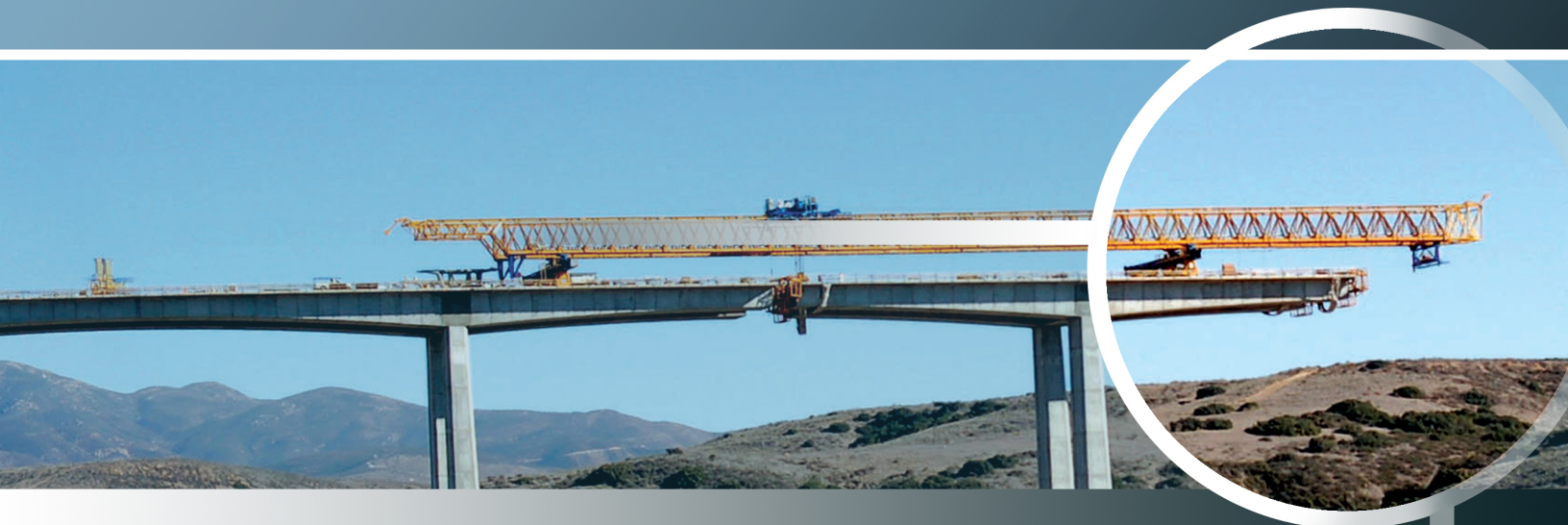




TRAINING MANUAL



AN INTRODUCTORY GUIDE FOR BRIDGE PROJECTS

LARSA 4D
THE COMPLETE SOFTWARE FOR BRIDGES

LARSA 4D Introductory Training Manual for Bridge Projects

A manual for

LARSA 4D
Finite Element Analysis and Design Software

Last Revised February 2013

Copyright (C) 2001-2012 LARSA, Inc. All rights reserved. Information in this document is subject to change without notice and does not represent a commitment on the part of LARSA, Inc. The software described in this document is furnished under a license or nondisclosure agreement. No part of the documentation may be reproduced or transmitted in any form or by any means, electronic or mechanical including photocopying, recording, or information storage or retrieval systems, for any purpose without the express written permission of LARSA, Inc.

Table of Contents

Introduction	5
About This Manual	5
Getting Started	7
Properties	9
Material Properties	9
Cross-Section Properties	9
Geometry	11
Joints	11
Members	11
Pier Bearing	12
Abutment Bearings	13
Refining the Model	16
Self-Weight Dead Load	17
Linear Static Analysis	19
Influence Line Live Load Analysis	23
Preparing the Influence Coefficients: The Lane	23
Preparing the Influence Coefficients: The Load Case	25
Vehicle and UDL Setup	26
Accessing Results	28
Combining the Left and Right Lanes	30
Staged Construction Analysis	33
Structure Groups	33
Construction Loads	34
Construction Stages	35
Running the Analysis	38
Getting Out of Staged Results	39
Tendons	41
Creating Tendon Geometry	41
Setting Tendon Properties	43
Stressing the Tendon	45
Tendon Results	45
Time-Dependent Material Effects	47
Choosing the Design Code	47
Setting Material Properties	48
Casting and Construction Day	49
Running the Analysis	50

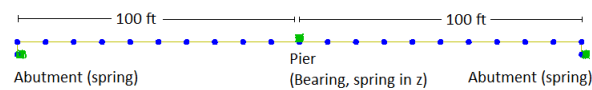
Results	51
Haunches	55
Creating a Non-prismatic Section	55
Spans	57
Simple Parabolic Variation of Depth	58
Piecewise Variation	58

Introduction

About This Manual

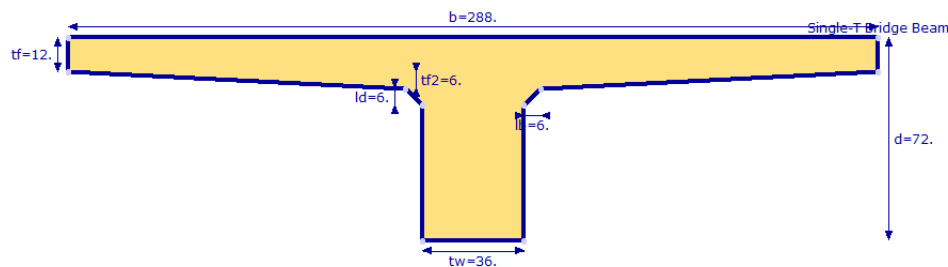
This tutorial provides an overview for using LARSA 4D for the analysis of a straight, two-span, single girder bridge. The bridge consists of a T-shape cross-section.

The example bridge in this guide consists of a T-shape cross-section. It has two spans of 100 feet each. The abutments are modeled as bearings supporting translation and rotation. The pier is modeled as a bearing supporting vertical displacement only.



A Two-Span Bridge

The cross-section will be a T-shape as given below:



Girder Cross-Section (units: inch)

Key Concepts

In the first half of this guide we will create the LARSA 4D model by:

- Setting project units
- Importing materials and custom parametric sections
- Creating joints in the spreadsheets
- Drawing members and creating members in the spreadsheets
- Creating bearings using simple linear grounded springs and 6x6 stiffness matrix definitions
- Using rigid members
- Refining the model using Break Members
- Creating a self-weight load case
- Viewing the results of a linear static analysis with graphical results and results spreadsheets

In the second half of this guide we will cover several analysis techniques:

- Live load analysis for AASHTO LRFD with influence lines
- Structure Groups
- Staged Construction Analysis
- Temporary Loads
- Support Change Activities
- Post-Tensioning with tendons
- Material time effects under CEB-FIP 90
- Creating haunches with non-prismatic cross-section variation

About LARSA 4D

LARSA 4D is an advanced multipurpose 3D structural analysis package featuring a powerful graphical user interface and an analysis engine with unmatched analytical features including influence line and surface based live load analysis, staged construction analysis, time-dependent material properties and segmental construction analysis, hysteretic and seismic elements and seismic analysis, and progressive collapse.

The LARSA structural analysis engine has been in commercial use for over 25 years. It was originally developed to perform nonlinear static analysis of structures that have large displacements, such as suspension and cable stayed bridges and guyed towers. The engine became popular for analyses of these types of structures because of its unprecedented accuracy at a reasonable price. The engine has been powerful since day one, using both tangent stiffness and the full Newton-Raphson method with iterations in nonlinear analysis. LARSA software has come a long way since it was first available on the VAX super-mini computers decades ago.

LARSA, Inc. has always been an industry leader. LARSA was the first to offer an individual PC-based DOS structural analysis package with geometric nonlinear analysis capabilities in 1986. In 1994, LARSA took the early next step to Microsoft Windows with a point-and-click graphical user interface and two years later was the first to offer elastic/perfectly plastic pushover analysis. Today, LARSA's flagship product is LARSA 4D, released in 2006 and featuring new seismic and inelastic elements, major improvements to influence and staged construction analysis, and many new features for bridge design and analysis.

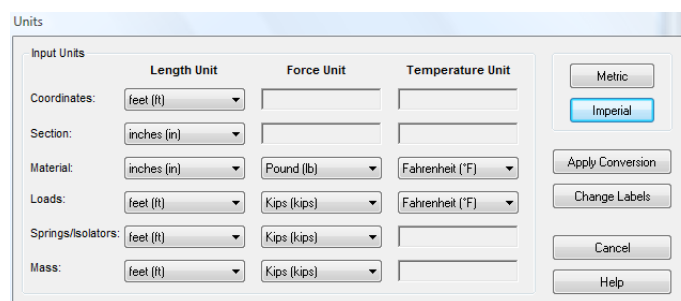
Getting Started

We will now begin the tutorial.

- ☛ Open LARSA 4D, or if LARSA 4D is already open start a new project.
- ☛ Save the project, such as with the name “basic bridge example”.

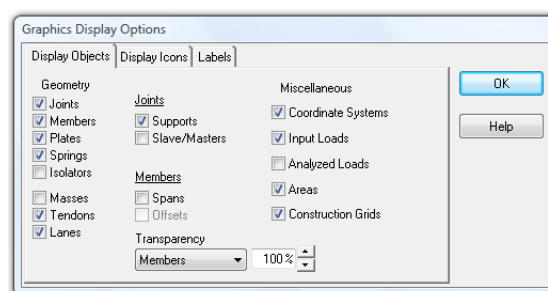
Units in LARSA 4D come in six categories: coordinates, sections, materials, loads, springs/isolators, mass elements. Each category can make use of different units for convenience. Coordinates are often entered in meters or feet, while sections are often specified in inches or centimeters. The Units window is arranged in a grid, with the unit categories arranged vertically and the unit types (length, force, temperature) horizontally.

- ☛ Set the units to be used in the project with the **Input Data** → **Units** command. The units should be as indicated in the figure below. If any changes need to be made, choose **Apply Conversion**.



Units

- ☛ Set up initial visual display options in the **Graphics** → **Show** command, which controls which aspects of the project model are included in the graphics view. Be sure Joints, Members, Springs, Tendons, Lanes, and Supports are turned on.



The Show Command

For More Information, please refer to the following documentation.

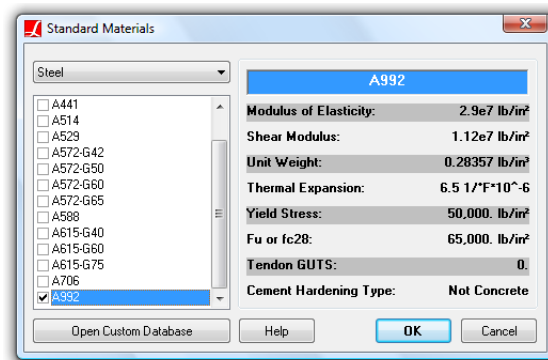
- Model Units in *LARSA 4D User's Manual*.
- Graphics Display Options in *LARSA 4D User's Manual*.

Properties

Before starting on the geometry of the project, we will bring in material and section properties to assign to the girder and, later, the post-tensioned tendon.

Material Properties

- ☛ In the **Input Data** menu click the **Materials** tool. It is shown below.



Importing Standard Materials

The materials tool presents a database of common material properties. They are not available for use until they are imported into the project.

- ☛ Make the Steel category selected and choose a steel material appropriate for the strands of a tendon, such as A992. Click the material so it is checked.
- ☛ Change to the Concrete category and choose a material appropriate for the girder section, such as Fc_4. Click the material so it is checked.
- ☛ Click **OK** to import the materials.

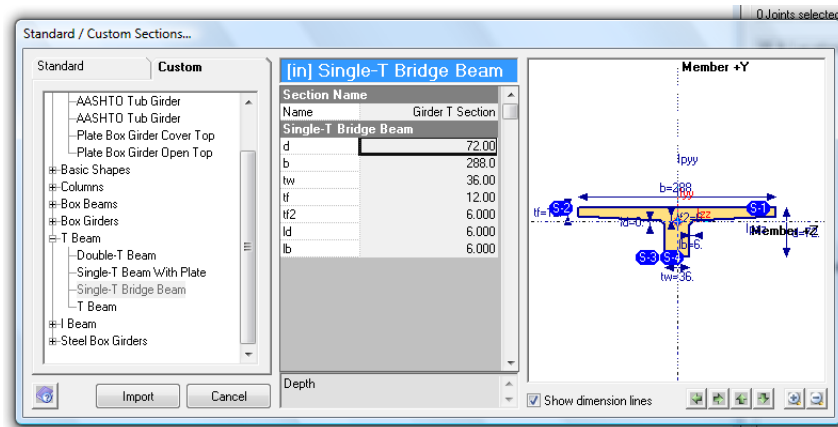
Cross-Section Properties

We will use a parametric cross-section shape for the T-section, which comes in LARSA 4D's custom section shape library.

- ☛ Go to **Input Data** → **Sections**.
- ☛ Click the **Custom** tab.
- ☛ Find the T Beam group, expand the group by double-clicking it, and choose Single-T Bridge Beam.

The shape is customized by setting its name and seven dimension parameters.

- ☛ Set the name and parameters according to the figure below.



Importing a custom T-shape section

- Click **Import** to bring the section definition into the project.

LARSA 4D will ask whether it is OK to create a new parametric section database for the model in which to put the section definition.

- Click OK and save the file along with your project. You may use the file name “basic bridge example sections”.

Remember where you save the file! You will need to find it in the next section. It will have a LPSX file extension.

For More Information, please refer to the following documentation.

- Properties in *LARSA 4D Reference Manual*.

Geometry

Because this model is of a straight bridge, we will lay out the geometry along the global x-axis starting at 0. The pier will be at 100 feet. The final abutment will be at 200 feet. The elevation axis will be the global z-axis.

There are several ways to enter geometry in LARSA 4D: using spreadsheets, the Model Data Explorer, drawing the geometry graphically with the mouse, the generation and transformation tools, and importing geometry from other programs such as Excel or CAD software. We will start by using the spreadsheets in this tutorial.

Model geometry in this tutorial consists of three types of objects: joints, members, and grounded springs. Joints represent connection points between elements and have six degrees of freedom. Members are two-node line elements. They will be used to represent the girder. Grounded springs are one-node spring elements used to model soil-structure interaction or connections between bridge superstructure and piers or abutments at bearings.

Joints

- ☛ Open the geometry spreadsheets using **Input Data** → **Geometry**. The Joints spreadsheet is the first spreadsheet that opens in the Geometry group.
- ☛ In the blank row, enter the coordinates (0, 0, 0). After the first value is entered, the row becomes initialized as a joint.

Spreadsheets always have a final blank row which is used to enter more data.

- ☛ Repeat this to enter joints at (100, 0, 0) and (200, 0, 0).

The structure will be supported by bearings, which will be modeled as a grounded spring elements later on. As a result, we do not need to set any translation or rotation restraints here.

	ID	X (ft)	Y (ft)	Z (ft)	Translation DOF	Rotation DOF	Displacement UCS
1	1	0.0000	0.0000	0.0000	all free	all free	Global
2	2	100.0000	0.0000	0.0000	all free	all free	Global
3	3	200.0000	0.0000	0.0000	all free	all free	Global
4							

Joints Spreadsheet

Members

To create the girder, it is also possible to use a spreadsheet. We will instead use the drawing tool to show more of the capabilities of LARSA 4D.

- ☛ Change to the **Graphics View** window by clicking the window button at the bottom of the screen. Then click the toolbar button for Zoom Extents.

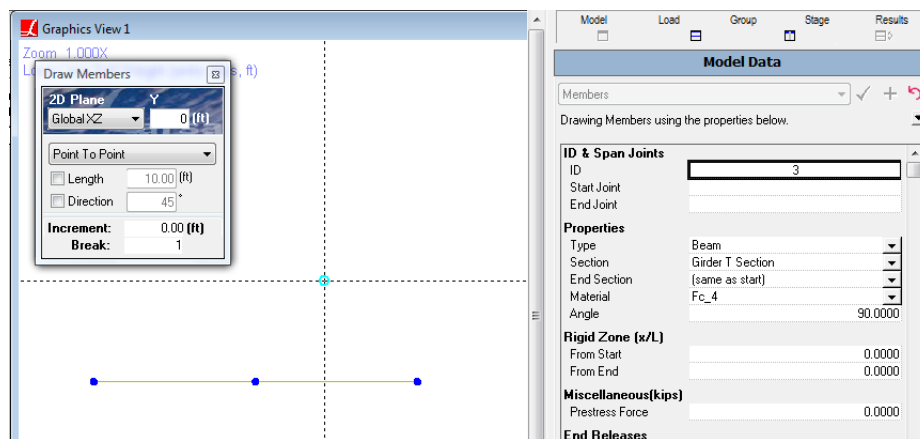
The drawing command allows us to click the joints where members should go.

- ☛ In the **Draw** menu, activate **Geometry** → **Members**.
- ☛ In the Model Data Explorer on the right-hand side of the screen, choose the appropriate section (“Girder T Section”) and material (“Fc_4”) for the girder. Also set orientation angle to 90 degrees, which is the usual convention for horizontal members.

No other fields in the Model Data Explorer need to be set now.

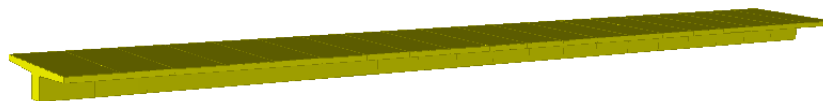
- ☛ Click the leftmost joint (and then release the mouse button) to start drawing a member from that joint. Then click the middle joint to complete drawing the member for the first span.
- ☛ Click the middle joint and then the rightmost joint to create the member for the second span.

The members are created with the section, material, and orientation angle we chose.



Drawing Members

- ☛ To end the drawing operation, close the floating Draw Members window by clicking on its X close button.
- ☛ Check that the members have been set up correctly using **Graphics** → **Complete Rendering**. You may want to use the graphics rotation tools to get a better view.



Complete rendering of the model so far

- ☛ Turn off complete rendering using **Graphics** → **Simple Rendering** and if necessary return the graphics view to a side view.

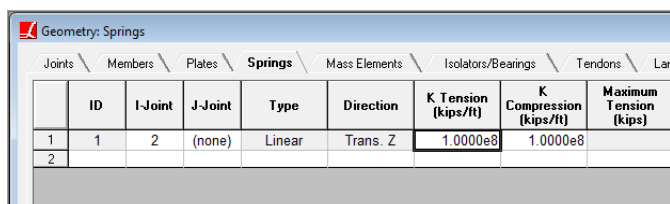
Pier Bearing

The pier will be modeled as a bearing (a grounded spring) acting in the z (elevation) direction only.

Although the pier joint is located at the girder centroid and the bearing only connects at the bottom of the T shape, because the bearing acts in the z direction only its vertical position is not relevant. We will pay closer attention to the bearings at the abutments.

- Return to the geometry spreadsheets and change to the **Springs** tab.
- Add a new row to the spreadsheet using the **Add Row** tool in the toolbar or in the **Edit** menu. Or just start typing into the blank row.
- In the I-Joint cell, enter joint number 2. Leave the J-Joint cell empty, signifying that this is a one-node (i.e. grounded) spring.
- Change the spring direction to Trans. Z and set its K Tension value to 1e8. (The K Compression cell updates to have the same value since a linear spring must have equal stiffness in both tension and compression.)

Spring directions are in joint displacement coordinate system directions. All of our joints in this tutorial use the Global Coordinate System as their displacement coordinate system, and so we have set this spring to act in the vertical direction.



	ID	I-Joint	J-Joint	Type	Direction	K Tension (kips/ft)	K Compression (kips/ft)	Maximum Tension (kips)
1	1	2	(none)	Linear	Trans. Z	1.0000e8	1.0000e8	
2								

Adding a Bearing

Abutment Bearings

The abutments will also be supported using bearings. The bearings at the abutments will restrict the deformation of the bridge in all directions of translation as well as in the direction of axial rotation.

It is important to make sure that any model has no rigid body motions by including enough restraints to prevent uncontrolled motion. The restraints described so far are enough to prevent rigid body motion.

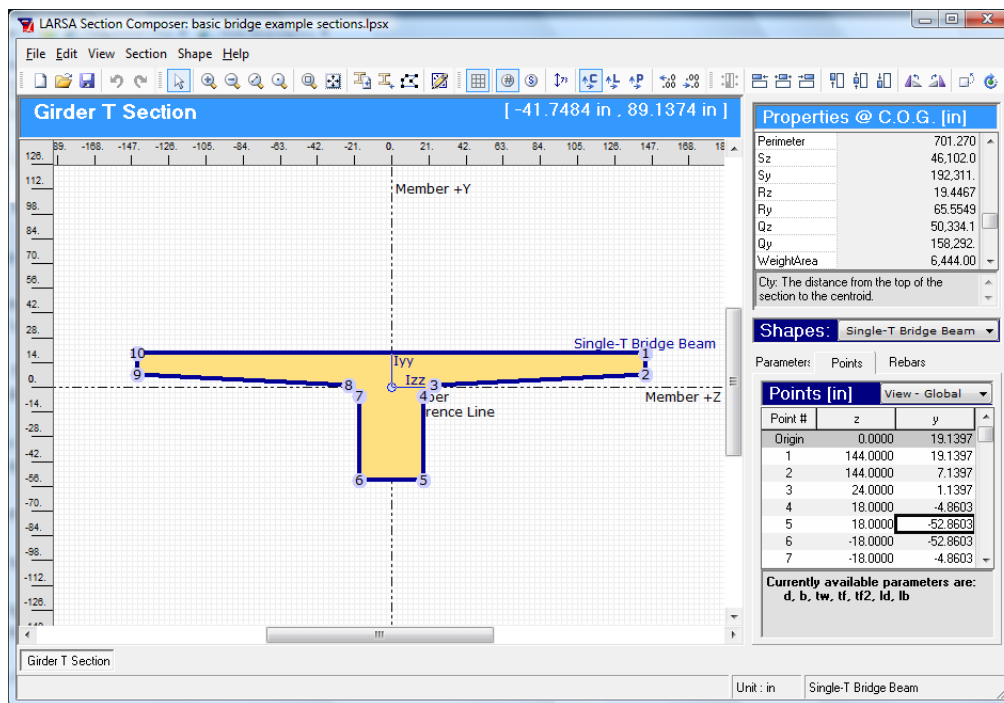
The bearings are connected at the bottom of the girder's web. However, the joints in the model have been placed at the girder's centroid. It is necessary to place a joint at the bottom of the web and connect it to the centroid using a rigid member element. We can find out the distance from the centroid to the bottom of the web using LARSA Section Composer.

- Start LARSA Section Composer and open the database created by LARSA 4D earlier ("basic bridge example sections"). You should leave LARSA 4D running in the background.
- Click inside the T shape to activate it.
- In the Shapes explorer in the right side of the screen, change to the **Points** tabs. Then change "Edit - Local" to "View - Global".

The "View - Global" option shows the location of the points on the perimeter of the shape in the member reference coordinate system, which in this case is aligned with the member centroid.

- Note the direction of the member reference axes in the graphics view of the cross-section. Member +Y is up. (The member reference axes differ from the global coordinate system axes.)

- Make sure **Show Point Numbers** is turned on in the **View** menu. Then identify a point number at the bottom of the web (point 5 or 6) and in the points spreadsheet note the points' y-coordinate (-52.8603 in).



Use LARSA Section Composer to find the location of the bottom of the T section relative to the centroid

- Close Section Composer.
- Back in LARSA 4D, return to the joints spreadsheet by clicking the **Joints** tab.

The member reference y-axis corresponds to the global coordinate system z-axis. That is because the member's orientation angle was set to 90 degrees. Also note that the joint coordinates units is feet in the spreadsheet, not inches as shown in Section Composer.

- Add a new row to the spreadsheet.
- In the new row's Z cell, type " $= -52.9/12$ " (without the quotes). Simple formulas can be entered into LARSA 4D spreadsheets (although unlike in Excel the formulas are not saved). The value -4.4083 will be computed and then entered into the spreadsheet. This creates a joint at (0, 0, -4.4083).
- Add another row to the spreadsheet to create a joint at (200, 0, -4.4083).

Joints							
Joints Members Plates Springs Mass Elements Isolators/Bearings Tendons							
Joints Slave/Masters Rigid Links Rigid Diaphragms							
	ID	X (ft)	Y (ft)	Z (ft)	Translation DOF	Rotation DOF	Displacement UCS
1	1	0.0000	0.0000	0.0000	all free	all free	Global
2	2	100.0000	0.0000	0.0000	all free	all free	Global
3	3	200.0000	0.0000	0.0000	all free	all free	Global
4	4	0.0000	0.0000	-4.4083	all free	all free	Global
5	5	200.0000	0.0000	-4.4083	all free	all free	Global
6							

The joints spreadsheet after adding the bearing joints

The two new joints 4 and 5 must be connected to the joints at the centroid (joints 1 and 3) using rigid connections.

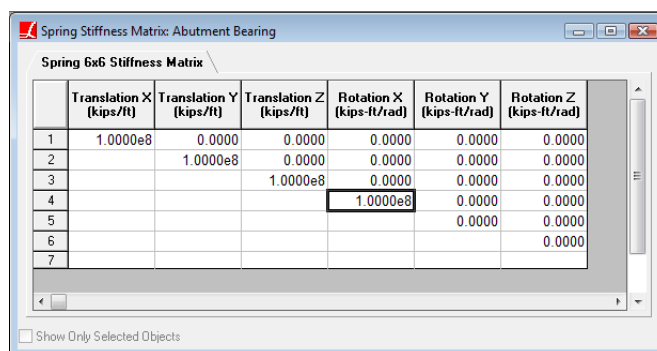
- ◆ Change to the **Rigid Links** sub-tab.
- ◆ Add two rows to the spreadsheet, and connect joints 4 to 1 and joints 5 to 3.

If bearings act in multiple directions, it is possible to put multiple grounded springs at a single joint each acting in one direction, or a single grounded spring with a 6x6 stiffness matrix can be used to define its stiffness in all directions at once. We will use a 6x6 stiffness matrix.

- ◆ Open the **Properties** spreadsheets from the **Input Data** menu and change to the **Spring Properties** tab.
- ◆ Add a new row to the spreadsheet. Name the new spring property definition “Abutment Bearing”. Change its type to 6x6 Stiffness Matrix.
- ◆ Then right-click the row and choose **Edit Stiffness Matrix**.

The spreadsheet that appears allows you to edit the 6x6 stiffness matrix for this element. Stiffness matrices are symmetric matrices that represent the relation between displacement and force. Each cell represents essentially a spring constant relating one direction of displacement with one direction of force. For basic springs, only the diagonal cells are used. The off-diagonal cells are left as zero. Because the matrix must be symmetric, the cells below the diagonal cannot be edited. The diagonal cells are the bottom-most editable cells.

- ◆ In the three Translation cells on the diagonal and the Rotation X cell on the diagonal, enter 1e8.



	Translation X (kips/ft)	Translation Y (kips/ft)	Translation Z (kips/ft)	Rotation X (kips-ft/rad)	Rotation Y (kips-ft/rad)	Rotation Z (kips-ft/rad)
1	1.0000e8	0.0000	0.0000	0.0000	0.0000	0.0000
2		1.0000e8	0.0000	0.0000	0.0000	0.0000
3			1.0000e8	0.0000	0.0000	0.0000
4				1.0000e8	0.0000	0.0000
5					0.0000	0.0000
6						0.0000
7						

6x6 Stiffness Matrix for the Abutment Bearings

- ◆ Close the “Spring Stiffness Matrix: Abutment Bearing” window. Then close the Properties spreadsheets.

We will now create two more springs and assign the 6x6 stiffness matrix to the bearings.

- ◆ In the Geometry spreadsheets, change to the **Springs** tab. The bearing we created previously will still be there.
- ◆ Add two more springs, at joints 4 and 5. Ignore the K Tension field this time and instead scroll far to the right and set the Properties Definition fields to Abutment Bearing for both new bearings.

Location	K Tension (kips/ft)	K Compression (kips/ft)	Maximum Tension (kips)	Maximum Compression (kips)	Hook (ft)	Gap (ft)	Properties Definition
ns. Z	1.0000e8	1.0000e8					(none)
6x6)	0.0000	0.0000					Abutment Bearing
6x6)	0.0000	0.0000					Abutment Bearing

Adding the Abutment Bearings

Close the Geometry spreadsheets window. The graphics window should remain.



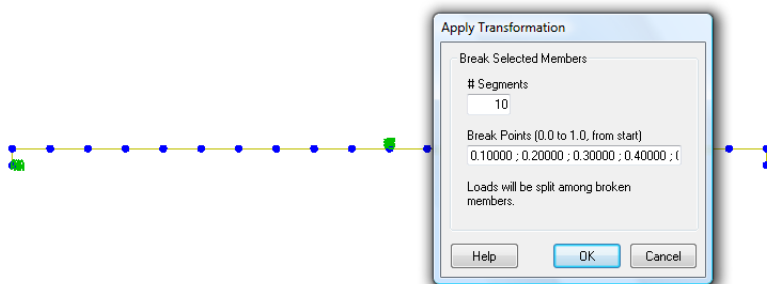
The model after the bearings have been added

Refining the Model

Before running an analysis, it is important to break up long member elements into small pieces. Displacements are computed only at the locations of joints in the model, so it is necessary to put joints at intermediate locations on each span.

We will use the Break Members tool to break the span members into smaller pieces. The Break Members tool operates on selected geometry (only).

Use the **Modify** → **Break Members** tool to break the two selected members in the model into 10 pieces each. Enter 10 for the number of segments and then click OK.



Breaking Members

For More Information, please refer to the following documentation.

- For model geometry reference, see Geometry in *LARSA 4D Reference Manual*.
- For more on using spreadsheets, see Using the Model Spreadsheets in *LARSA 4D User's Manual*.
- For help with drawing commands, see Drawing Geometry and Loads in *LARSA 4D User's Manual*.
- LARSA Section Composer in *LARSA Section Composer Manual*.
- Spring Property Definitions in *LARSA 4D Reference Manual*.

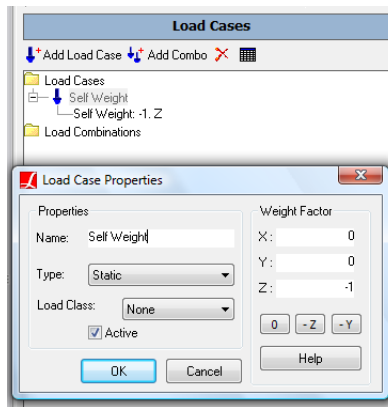
Self-Weight Dead Load

In this section we will create a static load case for self-weight only. The Load Cases Explorer is the primary way to create load cases.

- Click the **Load** button above the Model Data Explorer to activate the Load Cases Explorer.

The Load Cases Explorer will take the place of the Model Data Explorer.

- Add a new load case by clicking the Add Load Case button at the top of the explorer.
- Right-click the load case and choose **Properties**. Change its name to “Self Weight”. Then enter -1 in the Weight Factor Z field to indicate that gravity applies in the negative-z direction with a factor of one. (The **-Z** button enters this value for you.)
- Click **OK** to finish changing the properties of this load case.



Self Weight Load Case

For More Information, please refer to the following documentation.

- Self Weight in *LARSA 4D Reference Manual*.

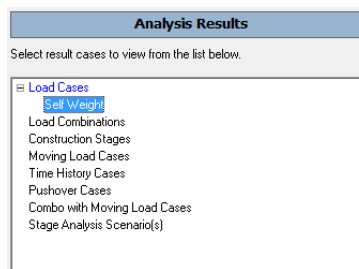
Linear Static Analysis

At this point we will run a linear static analysis. The analysis will compute the effects of the self-weight load case created in the previous section.

- ☛ From the **Analysis** menu, choose **Linear Static / P-Delta Analysis**.
- ☛ There are no options for this analysis type. Click **Analyze**. You will be prompted to save the project and to compute the torsion constant for the cross-section definition. Do so.
- ☛ Once the analysis completes successfully, close the analysis window.

The Analysis Results Explorer will be automatically opened on the right side of the screen.

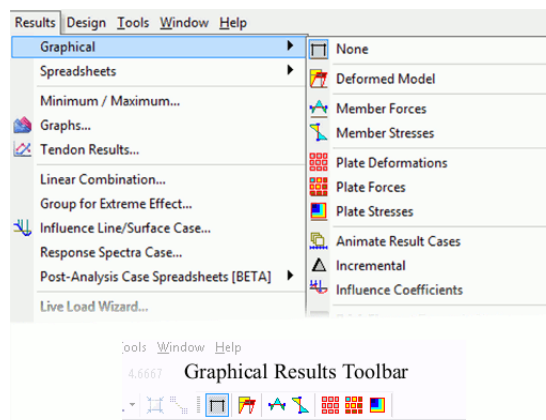
- ☛ Set the results units to the Imperial defaults (inches for displacement and kips for forces) in **Results** → **Units**.
- ☛ Open the Load Cases group and select the Self Weight result case from the Analysis Results Explorer.



Select the Self Weight Result Case

There are several ways to examine analysis results in LARSA 4D, including graphics and spreadsheets. We will demonstrate graphical results first.

- ☛ Turn on **Deformed Model** graphical results either from the **Results** menu or the toolbar.



Graphical Results Menu and Toolbar

- ☛ Select all elements using **Selection** → **Select Objects** → **Select All**.

Results are only shown for selected elements.

- ☛ Drag up the scale factor slider until you can see deformation. The slider is at the bottom of the Analysis Results Explorer.



Graphical Results Scale Factor

Reaction labels can be turned on from the floating Graphical Results Options tool window.



Deformed Model

- ☛ Turn off graphical results by choosing **None** in the graphical results menu or toolbar.
- ☛ Open the stresses spreadsheet using **Results** → **Spreadsheets** → **Member** → **Stresses**.

The stresses spreadsheet shows stresses at pre-set locations on the perimeter of the cross-section called stress recovery points. Four stress recovery points have been pre-set for the T-shape at the four extreme corners.

Stresses are shown at segments within each member element. The number of segments to divide each member into is controlled in **Results** → **Results Display Settings**.

Member Stresses								
☐ Envelope Selected Result Cases			☐ Incremental Results					
	Member	Station	Result Case	(P/A) @ Centroid (kips/in ²)	Normal Stress @ Point 1	Normal Stress @ Point 2	Normal Stress @ Point 3	Normal Stress @ Point 4
1	1	0	Load Cases:	-0.0346	0.0581	0.0581	-0.2908	-0.2908
2	1	1	Load Cases:	-0.0346	0.0092	0.0092	-0.1558	-0.1558
3	1	2	Load Cases:	-0.0346	-0.0371	-0.0371	-0.0278	-0.0278
4	1	3	Load Cases:	-0.0346	-0.0810	-0.0810	0.0933	0.0933
5	1	4	Load Cases:	-0.0346	-0.1223	-0.1223	0.2073	0.2073
6	1	5	Load Cases:	-0.0346	-0.1610	-0.1610	0.3144	0.3144
7	2	0	Load Cases:	-0.0346	0.7136	0.7136	-2.1013	-2.1013
8	2	1	Load Cases:	-0.0346	0.6385	0.6385	-1.8938	-1.8938
9	2	2	Load Cases:	-0.0346	0.5659	0.5659	-1.6934	-1.6934
10	2	3	Load Cases:	-0.0346	0.4959	0.4959	-1.4999	-1.4999

Member Stresses

- ☛ Close the spreadsheet window before going on.

For More Information, please refer to the following documentation.

- Linear Static Analysis in *LARSA 4D Reference Manual*.
- Analysis Results Explorer in *LARSA 4D User's Manual*.
- Viewing Results Graphically in *LARSA 4D User's Manual*.

- Results Spreadsheets in *LARSA 4D User's Manual*.

Influence Line Live Load Analysis

Influence line and surface analysis is an extension of the moving load analysis which uses the effects of a unit load to rapidly compute the effects of arbitrary complex load patterns on design lanes. Influence-based results are often necessary to comply with bridge design codes, such as the AASHTO LRFD, which requires finding the worst case scenario out of more possibilities than can be analyzed individually.

LARSA 4D supports both influence lines and influence surfaces. Influence lines is used on models where girders are represented by linear elements, whereas influence surfaces is used on models with a deck surface modeled as plate elements. In this section we will use influence line analysis to find the worst-case truck positioning according to AASHTO LRFD.

There are two parts to influence analysis. The first part determines how the influence coefficients (i.e. unit loads) are computed on a lane on the bridge. The second part specifies vehicle and uniform load options according to a design code.

Preparing the Influence Coefficients: The Lane

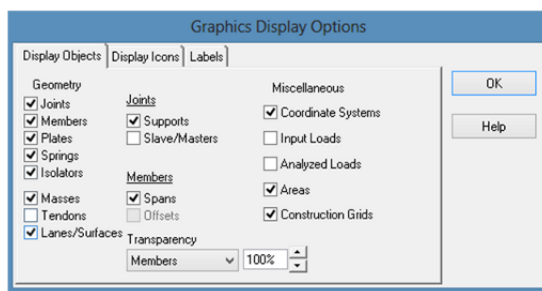
A “lane” in LARSA 4D is a linear path in the model along which a vehicle pattern or unit load will march in a moving load or live load analysis. A lane can take any path through 3D space. If it falls off of a girder, the loading will cause moments. For an influence line analysis, we must place a lane on the structure where a traffic lane will fall. Because our cross-section is 24 ft wide, two 12 ft lanes will fit: one 6 ft offset to the left (+y side) and one 6 ft offset to the right (-y side).

We will start by creating the right lane.

☛ Create a new lane using **Draw** → **From Selected Members** → **Lane**.

A new lane is created riding on the top of the cross-section surface. It appears as a line above the member elements.

☛ If you cannot see the lane, open **Graphics** → **Show** and ensure **Lanes/Surfaces** is checked.



Graphics > Show

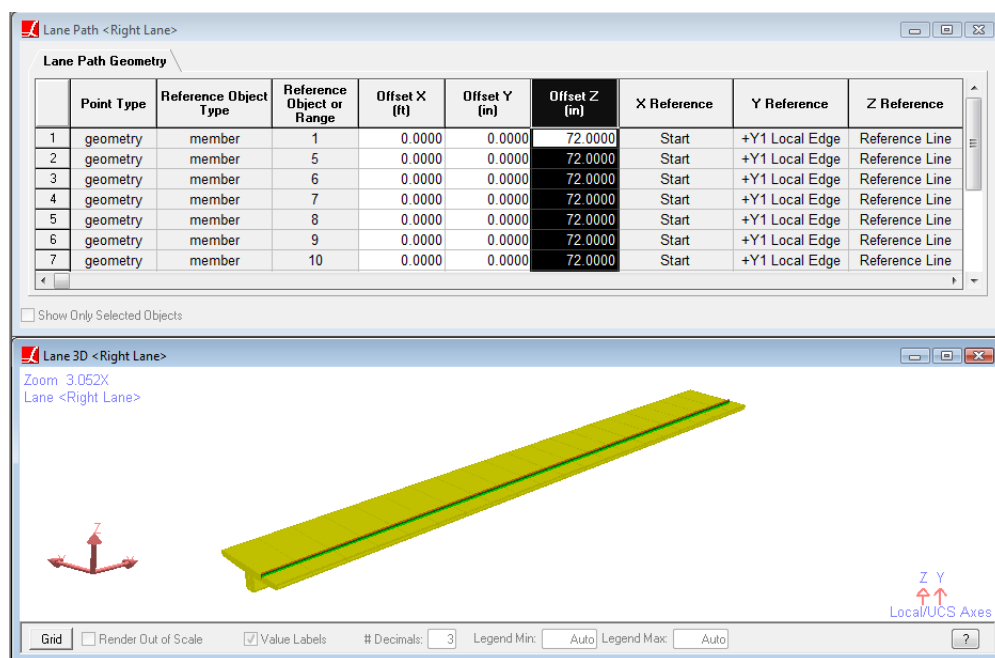
- Go to the Model Data Explorer by clicking **Model** above the explorers. Then switch to the Lanes panel (instead of Joints or Members, it should read Lanes at the top).
- Right-click the new lane and rename it to “Right Lane”.

We need to offset it by 6 ft in the global -y direction. The lane path is defined relative to the members the load is applied to, and offsets will be specified in local member directions. Global -y corresponds to member +z. The lane path is also specified in inches units (as per the initial choices in this tutorial), so the offset will be 72 inches in the +z direction, as you may recall from the display of the cross-section in Section Composer.

- Right-click the lane in the explorer and choose **Edit Path & View**.

A spreadsheet window on top for the path and a special lane graphics window below will open. Each row in the path spreadsheet specifies a control point on the lane path. The lane is made up of the straight line segments between the control points. (It is also possible to create a smoothly curved path.)

- Change the Offset Z to (positive) 72 for all rows. It will help to use the spreadsheet edit toolbar tool to change all of the cells at once.



Right Lane Setup

In order to comply with the part of the loading specification that says that in the two-trucks case each truck must be on a different span, we must tell LARSA 4D where the break is between the spans. We do this using a span break marker.

Each row in the path spreadsheet is a geometry control point in 3D space. The row for member 2 creates a geometry control point at the start of member 2, which is at the pier. A span break marker in the next row indicates the previous geometry control point ends a span.

- Insert a row into the spreadsheet after the row for member 2 and change its point type to span mark.

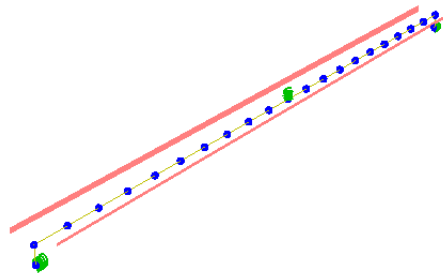
Lane/Surface Path Geometry						
	Point Type	Reference Object Type	Reference Object or Range	Offset X (ft)	Offset Y (in)	Offset Z (in)
10	geometry	member	11	0.0000	0.0000	72.0000
11	geometry	member	2	0.0000	0.0000	72.0000
12	span mark					
13	geometry	member	12	0.0000	0.0000	72.0000
14	geometry	member	13	0.0000	0.0000	72.0000

Lane Span Break Marker

- Close the lane spreadsheet. The lane graphics window will also automatically close.

This is the right lane. To create the left lane, we can duplicate the first lane definition and then shift it 12 feet to the left.

- Right-click the lane in the Model Data Explorer and choose **Duplicate Lane**. Rename the new lane “Left Lane”.
- Right-click the lane in the explorer and choose **Edit Path & View**. Change the Offset Z to -72 for all rows. Then close the lane spreadsheet.



Right and Left Lanes

Preparing the Influence Coefficients: The Load Case

The load case setup specifies additional options for how the lane is loaded, including the grid spacing of influence coefficients.

- In the Load Cases Explorer, add a new load case named “Influence Line Right”.
- Open the spreadsheets for the load case by double-clicking the load case. Then go to the **Moving Loads** tab, and then choose **Influence Loads** beneath the tabs.
- Add a new row to the spreadsheet. Change the lane column to Right Lane.
- Set the Forward Increment to 2 feet.
- Repeat the process to create a “Influence Line Left” load case. Then close the loads spreadsheet.
- Run a **Moving Load Analysis** from the **Analysis** menu. There are no options for this analysis. Just click **Analyze** and confirm to save the project. Close the analysis window when it completes successfully.

Vehicle and UDL Setup

The influence analysis in LARSA 4D was developed with AASHTO LRFD and AASHTO LFD in mind, although the influence solver can be used for many design specifications. We will use AASHTO LRFD in this example.

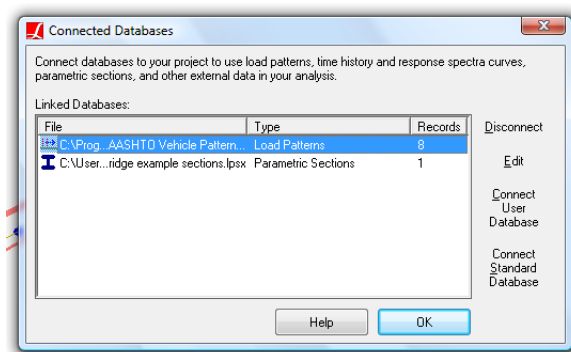
In the AASHTO LRFD code, a lane is loaded with combinations of a Design Tandem, Design Truck, and a Design Lane Load. LARSA 4D's standard load pattern database consists of the following two load patterns suitable for this code:

- HL-93 Design Truck in conformance with AASHTO LRFD 3.6.1.2.2. This truck has three axles of 8, 32, and 32 kips spaced (a minimum of) 14 ft. apart. The spacing between the two 32 kip axles may vary between 14 ft. and 30 ft. to produce extreme force effects. (Loads are lumped on the centerline of the lane, so no transverse effects are considered.
- HL-93 Design Tandem in conformance with AASHTO LRFD 3.6.1.2.3. The design tandem consists of a pair of 25 kip axles spaced 4 ft. apart.

The uniform load (UDL) is specified separately from the vehicle type.

We first must load into the project the vehicle definitions from a database of vehicles provided with LARSA 4D.

- ☛ Go to **Input Data** → **Connect Databases** and choose **Connect Standard Database**.
- ☛ Open the file "AASHTO Vehicle Patterns.dml". Then click **OK**.



Connecting the AASHTO LRFD Load Patterns Database

Each lane will be configured separately. For each lane, we will create an Influence Line result case which specifies how to perform AASHTO LRFD live load.

- ☛ Click **Results** → **Influence Line/Surface Case** to start configuring a new influence-based result case.
- ☛ Make sure that "Influence Line Coefficients Right Lane" is chosen at the top.

All of the default options on the first tab are correct. Note that **Load for Extreme Force Effects** is turned on.

- ☛ Click **Vehicular Loading**.
- ☛ Click **Add Lane Type**. Then click the **Select Vehicle Pattern** entry in the list on the left.
- ☛ On the right, where it reads **(Select Pattern)**, click it and choose HL-93/HS20-44 Design Truck.

Next we will enter the specification for the tandem condition.

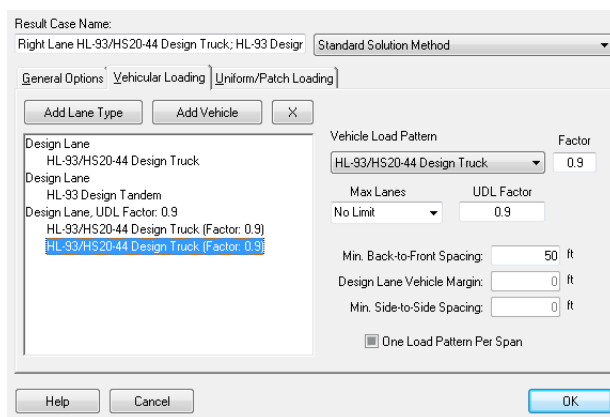
- Click **Add Lane Type**. Set the vehicle for this lane to HL-93 Design Tandem.

A Lane Type refers to the way a design lane may be loaded. The influence line solver will choose the lane type that produces the most extreme effects.

- Click **Add Lane Type** a third time. Set the (first) vehicle for this lane to HL-93/HS20-44 Design Truck. Set the factor (to the right of the vehicle type selection) to 0.9.
- Click **Add Vehicle**. A second HL-93/HS20-44 Design Truck with the same factor is added into this lane type.
- Set the UDL Factor to 0.9. This applies to the current lane type. (We have not yet set the UDL magnitude.)
- Set the minimum back-to-front spacing to 50 feet. And turn on the option **One Load Pattern Per Span**.

The One Load Pattern Per Span option pertains to the two-trucks lane type, in which the two trucks must be placed on different spans. The span break markers inserted into the lane path definition control what LARSA 4D knows to be the spans.

- Verify that your screen looks like the following figure:



Influence-Based Results Setup

- Change to the Uniform/Patch Loading tab, and enter a UDL magnitude of 0.64 kip/ft.
- At the top of the window, set the result case name to Right Lane.
- Click OK to finish creating this case.
- Then repeat this process for the left lane.

The Analysis Results Explorer will show a new group called Influence-Based with two new result cases inside.



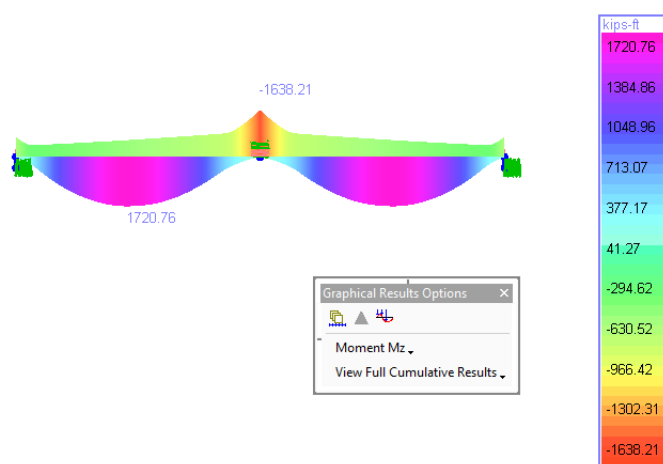
Influence Analysis Result Cases in the Analysis Results Explorer

Accessing Results

To access results, click the name of the case and use graphical or spreadsheet results. While each case individually does not provide the final values necessary for AASHTO LRFD (see below), they are the first step.

- Click the first influence case “Right Lane” in the Analysis Results Explorer.
- Turn on graphical member force diagrams and change the report to Moment Mz.

This graph represents the envelope of forces for all possible positions of the truck, tandem, or two trucks plus lane load, including variable axle spacing.



Moment Diagram

Accessing results in the spreadsheets works as usual, except you must choose an envelope column. In this case, you would choose the Mz column to envelope on. The rows of the spreadsheet then show the min and max Mz values for each station along the girder, plus corresponding forces in the other directions. Mz refers to member local coordinate system directions.

You may check where LARSA 4D decided to place the truck and lane load to produce the extreme effect for any point on the structure using several methods. The first method is to look at the Result Case column on the spreadsheet. It indicates the station number (in coordinate units) from the start of the lane at which each vehicle is placed (referring to the front of the vehicle).

Min/Max for Moment - Z - (kips-ft) Min/Max ABS					Sum Selected Cells			
Member	Joint	Result Case	Force X (kips)		Member	Moment Y (kips-ft)	Moment Z (kips-ft)	
1	1	Right Lane: s28.; s200.; UDL	-0.1465		062	0.0000	-0.6457	
2	1	Right Lane: s54.; s174.6; UDL	158.9257		368	0.0000	700.5923	
3	1	Right Lane: s170.4; UDL	-87.6273		409	0.0000	-425.9005	
4	1	Right Lane: s37.89; UDL	-77.2326		48.9911	0.0000	441.4382	0.0000 504.2109

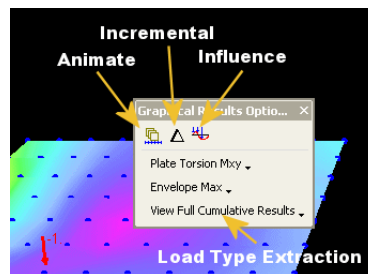
Numerical Influence Results and the Context Menu

It is also possible to right-click a row in an influence analysis spreadsheet and use either the **Create Input Load Case** or **Create Result Case** commands to create a new case that contains the loading configuration that produced the effect shown in the chosen row. Creating an input load case creates a new load case with member or plate loads for the vehicle axles and UDL. Creating a result case makes a new linear result combination out of the already-analyzed unit load result cases. It is then possible

to inspect either new case (or re-analyze an input load case) to obtain the static results for the loading configuration.

The graphical influence coefficient view is the recommended method to inspect the loading configuration that maximizes a result somewhere on the structure.

- ☛ Make sure you are at a graphics window with member sectional forces displayed.
- ☛ Click the Influence Coefficients button in the floating graphical results options tool window.



Influence Coefficients Mode

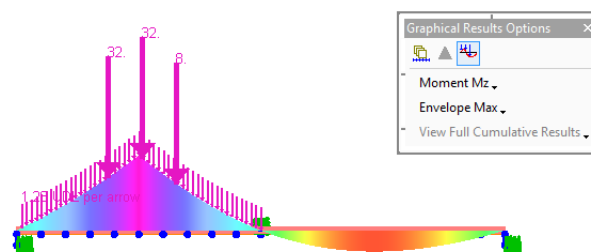
Once this tool is on, LARSA 4D is waiting for you to choose a location on the structure to maximize the forces at.

- ☛ Click any member to see influence coefficients for it. Be sure the Pointer tool is active.

You will be prompted to enter a station number for where you would like to see the coefficients. The stations range from 0 to the number of stations configured to be drawn graphically.

- ☛ Enter 0 to see the coefficients at the start of the member.

The graphics window updates to show the influence coefficients and the locations of axles at their worst position. Arrows indicate the locations of the axles of the vehicle specified by the active result case that produce the maximum moment at the chosen location. You can switch between placing the axles at the maximum and minimum (i.e. most negative) locations in the floating Graphical Results Options window.



Influence Coefficients View

- ☛ Turn off graphical results and close any open spreadsheets.

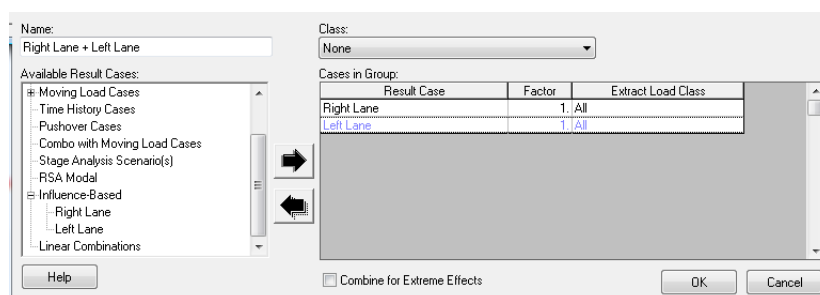
Combining the Left and Right Lanes

In order to compute the effects on the structure for AASHTO LRFD, we need to combine the effects of the left and right lane.

Because of multiple presence factors, there are three cases to consider. The right lane may be loaded individually with a multiple presence factor of 1.2. Similarly for the left lane. Or the two lanes may be loaded simultaneously with a multiple presence factor of 1.0.

We will create the simultaneous loading condition first. A Linear Result Combination simply sums the numerical results of two or more cases.

- ☛ Start the **Results** → **Linear Combination** tool.
- ☛ Add the two influence line cases (Left Lane, Right Lane) into the right list. Either double-click the influence cases or click a case and then click the big right arrow.
- ☛ Click **OK**.

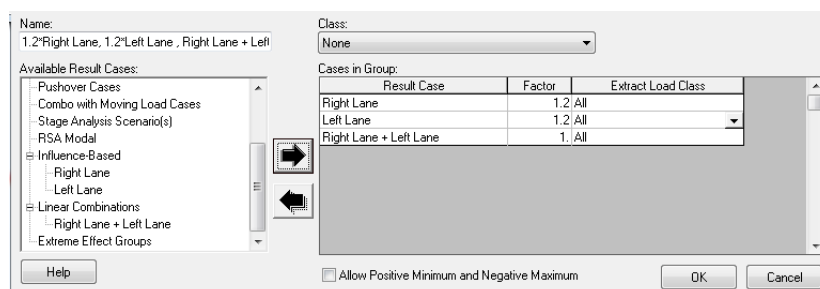


Linear Combination of Two Lanes

- ☛ Click the new linear combination case in the Analysis Results Explorer, and then open a results spreadsheet to view the combined effect from the two lanes.

Enveloping the three conditions (left, right, together) can be accomplished using Extreme Effect Groups which are like saved envelopes.

- ☛ Start the **Results** → **Group for Extreme Effect** tool.
- ☛ Find the three cases for the right lane, the left lane, and the new linear result combination and add them into the list on the right using the big right-arrow in the middle.
- ☛ Set the factors on each as appropriate. Then click **OK**.



Extreme Effect Group for Right Lane

You'll find the new groups in the Analysis Results Explorer in a new section called Extreme Effect Groups. You can access the results for these cases like any other case.

- Use the results spreadsheets with these cases to find the controlling effects on each lane.

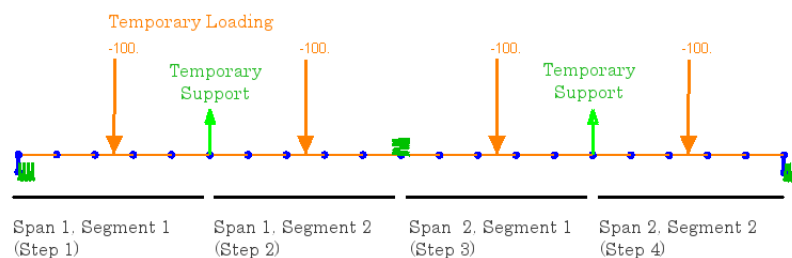
For More Information, please refer to the following documentation.

- Lanes in *LARSA 4D Reference Manual*.
- Influence Line & Surface Analysis in *LARSA 4D Reference Manual*.
- Influence Coefficients Graphical View in *LARSA 4D User's Manual*.
- Linear Result Combinations in *LARSA 4D User's Manual*.
- Extreme Effect Groups in *LARSA 4D User's Manual*.

Staged Construction Analysis

Staged Construction Analysis is a “4D” analysis. What is meant is that the analysis models changes to the structure over time. In this manual we will create a basic setup for staged construction using construction and loading activities. In this section we will set up the basic construction sequence. In the next section we will add a post-tensioned tendon and a tendon stressing activity.

The construction sequence will be in four steps from left to right. The first construction step will construct half of the first span with a temporary support at mid-span and a temporary construction load. The second construction step will assemble the second half of the first span, add a second temporary load, and remove the temporary support. The third construction step removes the temporary loads. This will then repeat for the second span.



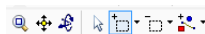
The Construction Sequence

Structure Groups

In order to run a Staged Construction Analysis, we will need to first define structure groups, which group the parts of the structure together that will be assembled together.

The first group will be the left half of the first span.

- ◆ Open the Structure Groups Explorer by clicking the **Group** button above the explorers.
- ◆ Unselect everything using **Selection** → **Unselect Objects** → **Unselect All** or any other method.
- ◆ Using the selection mouse tool (shown below), drag a window to select the grounded spring for the left abutment and the first five members on the girder.



Selection Tool

You should see the following after you have made the selection:

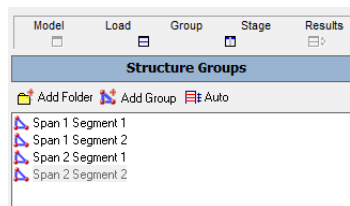


The First Structure Group

- Click the **Add Group** button in the Structure Groups Explorer to add a new group for the selected structural objects.

The new group automatically contains the currently selected members (and joints, but this will not be relevant).

- Click on the group and rename it to “Span 1 Segment 1”.
- Do the same for the remaining three segments: Unselect All, window select the segment, add a group, and then rename the group. Include the pier in Span 1 Segment 2, and include the right abutment in Span 2 Segment 2.



Structure Groups

- When you are done, select all geometry.

Construction Loads

Each condition of temporary or traveler construction loading must be defined as a load case. We will create four temporary loading conditions, which will be point loads at the mid-point of each girder segment. Each temporary load will be applied when the segment is first constructed and will be removed when the entire span is complete.

To identify the members we will place the temporary loads on, use the Pointer mouse tool. The Pointer tool is the fourth tool (the one to the left of the selection tool) in the figure above showing the graphics mouse tools.

- Turn on the Pointer tool and hover over the middle member in each segment. Hold the **CTRL** key while hovering the mouse over the members to see the member numbers, or look in the status bar at the bottom of the screen.

The members should be 4, 9, 13, and 18.

- Identify the two joints where we will need temporary supports (between the segments).

The joints should be 10 and 19.

An alternative way to identify member numbers is to turn on member number labels. This can be done with **Graphics** → **Show** or the keyboard shortcut **N**, which cycles through number label options. Press **N** once for joint numbers, and again for member numbers. Then press it again to cycle to spring numbers and again to turn off number labels.

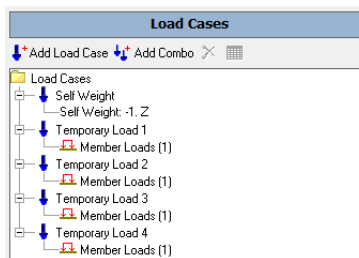
- ☛ Open the Load Cases Explorer by clicking **Load** above the Structure Groups Explorer. Then create four new load cases named “Temporary Load 1”, 2, 3, and 4.
- ☛ Double-click the first load case for temporary loading and switch to the **Member Loads** tab.
- ☛ Add a row to the spreadsheet. Set the member to 4, the direction to Global Z, and the magnitude to -100 (kips). Also change its start position to 0.5, meaning the middle of the member.

Joint Loads \ Support Disp \ Member Loads \ Plate Loads \ Moving Loads \ Time History \ RSA Loads							
Basic Loads		Thermal Loads					
	Member	Type	Direction	Magnitude at Start (kips or	Magnitude at End (kips or	Start Position (x/L)	End Position from start (x/L)
1	4	Point Force	Global Z	-100.0000		0.5000	
2							

Construction Loading: First Segment

- ☛ Click the second temporary load case and add the next member load (the same but on member 9). And then do the same for the third and fourth cases (members 13 and 18).
- ☛ Close the loads spreadsheet.

The Load Cases Explorer will indicate the type of loading in each case.



Construction Loading

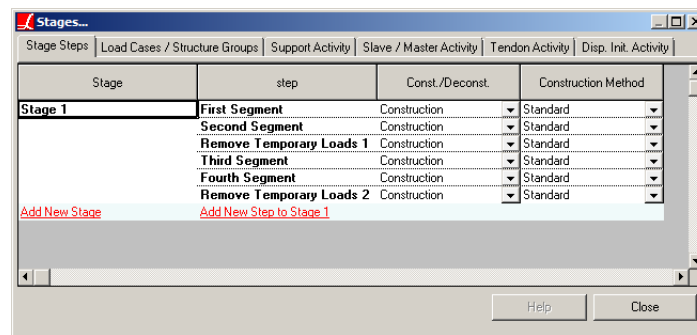
- ☛ Verify that the loading is correct in each case in the graphics window.

Construction Stages

Now we are ready to set up the staged construction stages and steps. A construction step is a set of construction activities that occur simultaneously. This will involve constructing part of the girder as well as applying loading. A construction stage consists of all of the construction steps that occur on a particular day of construction. Construction stages are used for time-dependent staged construction analysis where the day of construction matters. In this example, we will place all construction steps in a single construction stage.

- ☛ Go to **Input Data** → **Construction Stage Editor**.
- ☛ Double-click **Add New Stage** and leave the name as “Stage 1”.

- Then double-click **Add New Step to Stage 1** and change its name to “First Segment”. Do the same to create the next construction step for the second segment.
- After the first two steps, create a third step “Remove Temporary Loads 1”.
- Do the same to create the construction steps for the second span (the third segment, the fourth segment, and “Remove Temporary Loads 2”).



Construction Steps

The next step is to apply construction activities. The first step will include an activity to assemble the first five member elements.

- Change to the **Load Cases / Structure Groups** tab.
- Click the drop-down arrow next to **Add Group** besides the first construction step and choose “Span 1 Segment 1”.

Self-weight is not applied during Staged Construction Analysis unless it is explicitly included as an activity. A self-weight load case should be applied in any step in which elements are constructed, as in this case. Self-weight will be applied to all and only the newly constructed elements. (It should not be applied in other steps.)

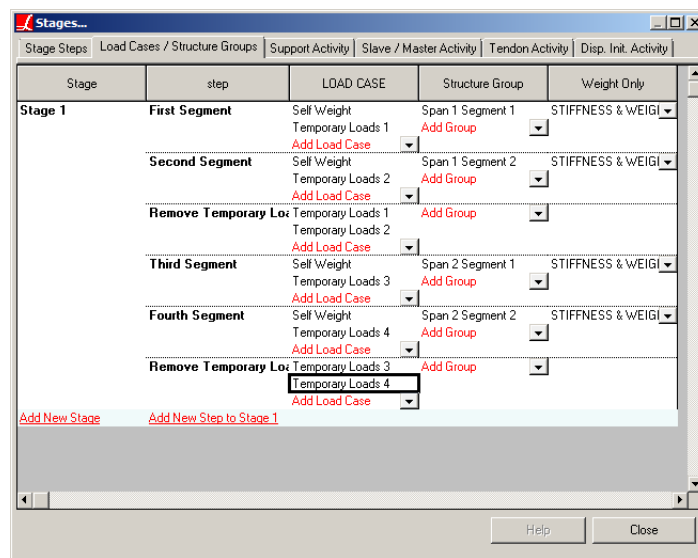
- Click the drop-down arrow next to **Add Load Case** besides the first construction step and choose “Self Weight”.

Simultaneously we will apply the first temporary load.

- Click the drop-down arrow next to **Add Load Case** below “Self Weight” and choose “Temporary Load 1”.
- For the Second Segment step, do the same: add the Span 1 Segment 2 group, self-weight (again), and Temporary Load 2.

To remove the temporary loading in “Remove Temporary Loading 1”, we will apply the same load cases again but with opposite sign. (When loading lasts just a single construction step, there is another way to make loads temporary. Rather than applying them with opposite sign in the next step, the step can be marked as containing temporary loading only.)

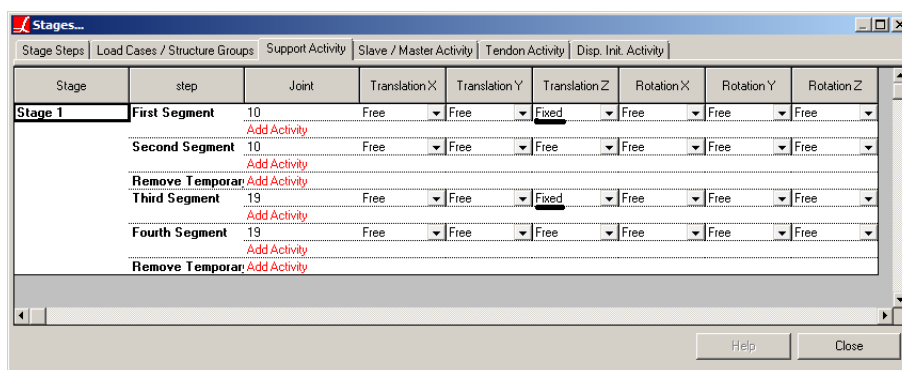
- Add Temporary Load 1 and Temporary Load 2 as load cases into the first removal step. We will reverse their sign later.
- Repeat this process for the second span.



Construction Stage Loading

Now we will add and remove temporary supports. In the First Segment step, we will add a temporary support at joint 9. In the Second Segment step, we will remove the support. The same will occur for joint 19 in the Third Segment and Fourth Segment steps.

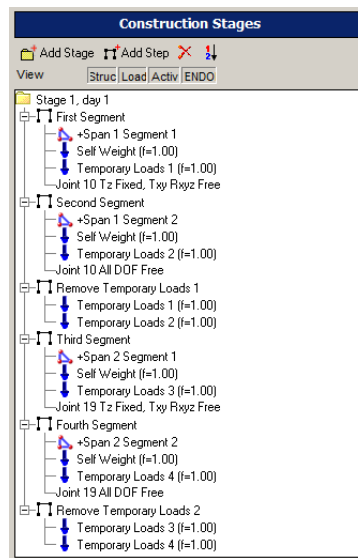
- ☛ Change to the **Support Activity** tab.
- ☛ Double-click the first **Add Activity** label and replace it with “10”. Press enter. Set its Translation Z to Fixed.
- ☛ Double-click the **Add Activity** label beside the Second Segment step and replace it with “10” (again). Press enter. This time its Translation Z will be free (the default).
- ☛ Do the same to fix Translation Z for joint 19 in the Third Segment step, and the free it in the Fourth Segment Step.



Temporary Support Activities

- ☛ Click **Close** when done.

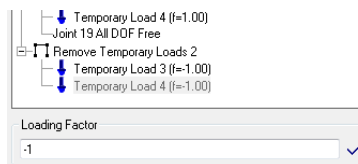
Look back in the Construction Stages Explorer to verify that the input is correct.



Construction Stage Loading

Notice the “(f=1.00)” next to each load case. This is a load factor. We will change these load factors to -1 to remove the temporary loading.

- ☛ Click “Temporary Loads 1” in the Remove Temporary Loads 1 step. At the bottom of the Construction Stages Explorer, where it says “No Property” in the figure it will now read “Loading Factor” and the field below it will have 1. Change this to -1 and click the checkmark to apply the change.
- ☛ Do the same to the three other load removal load cases.



Removing Temporary Loads

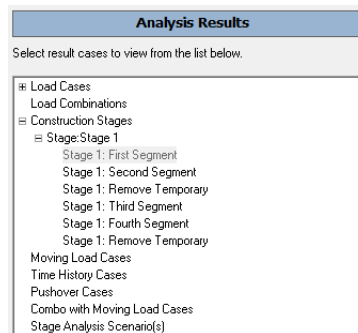
Running the Analysis

We can now run the Staged Construction Analysis. The analysis will compute the state of the structure after each construction step.

- ☛ Go to **Analysis** → **Staged Construction Analysis**. Click **Analyze**.
- ☛ After the analysis completes, close the analysis window.

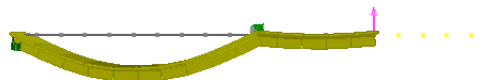
The Analysis Results Explorer will open automatically. This time the result cases are in the Construction Stages group. Inside this group there will be a group for each construction stage (we have just one), and inside that the results for each construction step.

- ☛ Open up the result groups and click the result case Stage 1: First Segment.



Staged Construction Results

- ☞ Turn on graphical deformation diagrams and set the scale factor to 128. Then step through the construction steps by clicking on them to observe the analysis sequence.

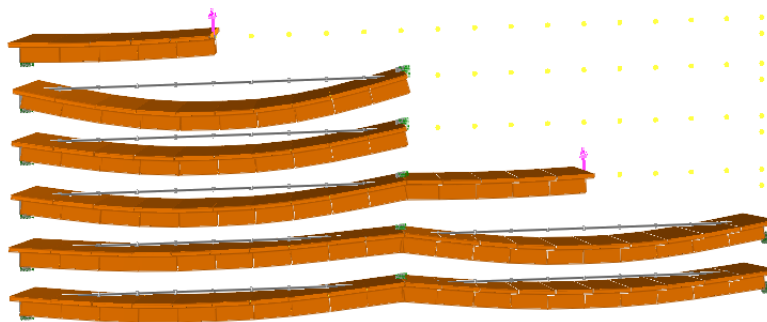


Staged Construction Results: Third Segment Step

- ☞ Turn off deformed model when you are done.

You can, of course, check the results numerically as well.

Using complete rendering, the construction sequence can be visualized by stepping through the analysis results:



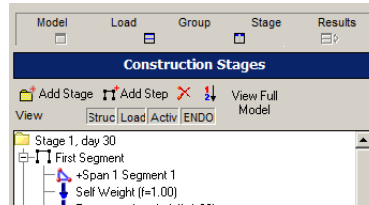
Deformation Results

Getting Out of Staged Results

As you step through the construction stage input or through staged construction results, you may find that the model starts to become grey or disappear. Elements that have not yet been constructed in the selected construction step or result case are shown greyed or, in some results and when complete rendering is turned on, hidden entirely.

This may become inconvenient as you move between results and revising model input. Or the graphics may become out of sync with the result case you are viewing.

When elements are greyed or hidden, a button appears in the Construction Stages Explorer called **View Full Model**. Click this button to reset the graphical view to show all elements, regardless of whether they have been constructed or not. If any stage or step is selected, it is unselected.



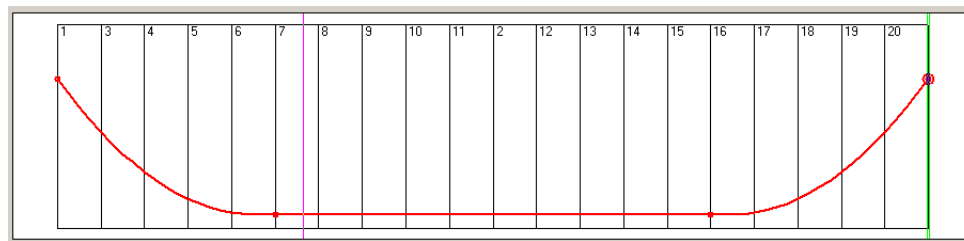
The View Full Model Button

For More Information, please refer to the following documentation.

- Structure Groups Explorer in *LARSA 4D User's Manual*.
- Loads in *LARSA 4D Reference Manual*.
- Staged Construction Analysis in *LARSA 4D Reference Manual*.

Tendons

Tendons are used to model pre- and post-tensioning of internal and external tendons. We will demonstrate a post-tensioned internal tendon in this tutorial with the following profile:



Final Tendon Profile

Creating Tendon Geometry

The easiest way to create the tendon path is to let LARSA 4D create one from the selected members. Then, the tendon can be fine tuned using the tendon path spreadsheet. This follows a similar process as creating a lane.

The only elements in this model are the members the tendon will pass through.

- ☛ Select All.

- ☛ Use the command **Draw** → **From Selected Members** → **Tendon**.

The Tendon Editor will open with a side view of the tendon. The tendon is currently a straight line running along the section centroid. We will alter the tendon so that it curves, reaching 5 inches above the bottom of the web at the mid span of the two spans (the start of members 7 and 16) and comes back up to the centroid at the end.

Tendons can take any path through 3D space. Geometry control points determine the path of a tendon. These control points are placed inside of members.

- ☛ Hold the **CTRL** key and click inside of member 7. Then do the same inside member 16.

This creates control points inside the two members. We will set their locations next.

- ☛ Click the first new control point, the one inside member 7.

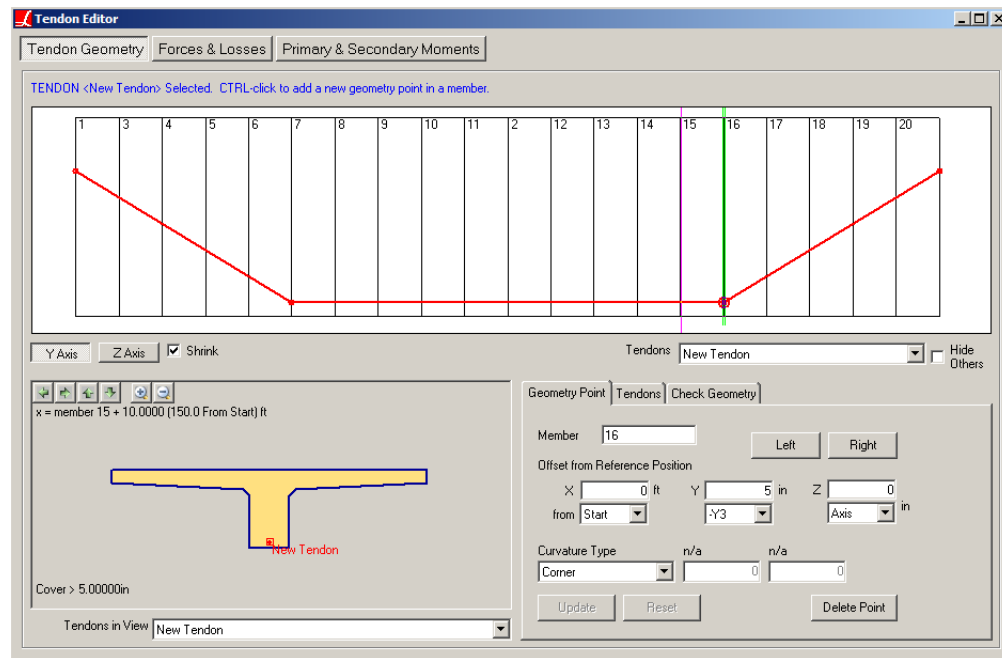
- ☛ In the lower right panel, change the X offset to zero, meaning the start of the member.

- ☛ Set the Y offset field to 5 and then change the reference from “Axis” (meaning the member reference axis, here the centroid) to “-Y3”

-Y3 signifies that the reference coordinate in the member is the third stress recovery point set in the cross-section definition. By convention, the third stress recovery point is at the bottom of the section, as it is in this case.

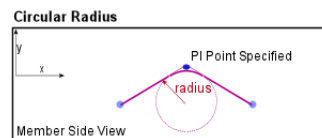
- ☛ Click **Update**.

- ☛ Do the same for the second new control point.



Tendon Editor

It is also possible to make the tendon smoothly curved. LARSA 4D has several tendon curve options. The first option creates a curve at a vertex on the path by rounding out the vertex with the arc of a circle of a given radius. After rounding out the vertex, the original geometry control point no longer falls on the tendon path. This is illustrated in the figure:

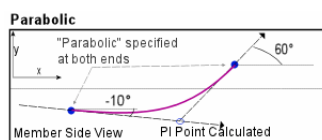


Tendon Circular Curve Fitting

- ☛ Click the first geometry control point we added, in member 7, to select it.
- ☛ At the lower right of the tendon editor, change curvature type from Corner to Circular Radius.
- ☛ Enter a radius of 200 (feet).
- ☛ Click **Update**.

Note how the tendon diagram has updated with the new curved path.

Another curvature method is to create a parabolic curve by specifying the tendon's direction (tangent) at the control points. This requires setting the curvature method to parabolic on the two geometry control points surrounding the curved segment. At each geometry control point, a Y Angle and Z Angle are set for the tendon's profile in the member's local y and z planes. The angle is entered in degrees, relative to the member's x-axis which is in this case, and is normally, the centroid.



Tendon ParabolicCurve Fitting

We will replace the circular curve with a parabolic curve.

- ☛ Click the very first geometry control point, in member 1, to select it. Change its curvature type from Corner to Parabolic.
- ☛ Enter a Y Angle of -10 (degrees).
- ☛ Click **Update**.
- ☛ Click the second geometry control point, in member 7, to select it. Change its curvature type from Circular Radius to Parabolic.
- ☛ Enter a Y Angle of 0 (degrees).
- ☛ Click **Update**.

We enter a Y Angle of zero for the second control point because the tendon is parallel to the member at this location. Notice the updated diagram with the smooth curve.

- ☛ Set up a similar curvature on the end of the tendon: Change the curvature type on the final two geometry control points to Parabolic. The Y Angle on the third point should be zero. The Y Angle on the final point should be (positive) 10. Be sure to click **Update** after changing each point.

The tendon now appears as in the first figure in this section.

There is also a spreadsheet for editing tendon geometry. In the Model Data Explorer or tendons geometry spreadsheet, right-click the tendon and choose **Path Spreadsheet**. The tendon path spreadsheet includes all of the same information as in the Tendon Editor. Additionally, however, you must specify every member that the tendon passes through. If the tendon passes through a member in which there is no need for a geometry control point, the member is added with the point type called path only.

TENDON Tendon 1									
Tendon Path Geometry									
	Point Type	Reference Object Type	Reference Object or Range	Offset X (ft)	Offset Y (in)	Offset Z (in)	X Reference	Y Reference	Z
1	geometry	member	1	0.0000	0.0000	0.0000	Start	Reference Line	Re
2	path only	member	3						
3	path only	member	4						
4	path only	member	5						
5	path only	member	6						
6	geometry	member	7	0.0000	5.0000	0.0000	Start	-Y3 Local Edge	Re
7	path only	member	8						

Tendon Path Spreadsheet

Setting Tendon Properties

- ☛ Keep the tendon editor window open, but also open the geometry spreadsheets (from the **Input Data** menu) and change to the Tendons spreadsheet.

- ☛ Rename the tendon to “Tendon 1” and set its properties as:

Material: A992
 Strand Area: 2 in²
 # of Strands: 3
 Jacking Force @ Start: 5500 kips
 Anchor Set: 0.15 in
 Wobble Coefficient: 0.0001 per ft
 Curvature Friction Coefficient: 0.15

- ☛ Close the tendon geometry spreadsheet.

The Tendon GUTS property of the steel material for the tendon must be entered.

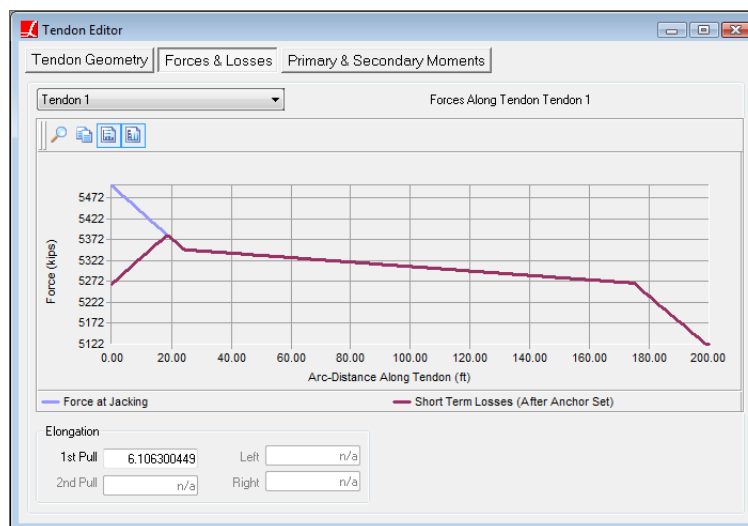
- ☛ Open **Input Data** → **Properties** → **Materials** → **More Properties**.

- ☛ Set the Tendon GUTS (guaranteed ultimate tensile strength) on A992 to 270,000 kips/in².

We can now look at the tendon force profile.

- ☛ In the Tendon Editor, change to the **Forces & Losses** tab to view the short-term force losses along the length of the tendon.

The forces and losses chart shows two curves superimposed. The top blue curve shows the forces in the tendon including the wobble and curvature friction coefficients but before anchor set has taken effect. The lower magenta curve shows the forces in the tendon after anchor set has occurred.



Tendon Short-Term Losses

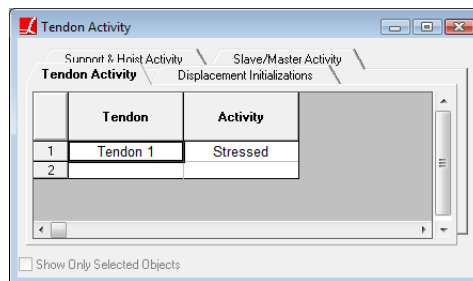
- ☛ Close the Tendon Editor.

Stressing the Tendon

The tendon will not affect the analysis until LARSA 4D is instructed to stress the tendon. Stressing tendons can be done as a construction activity in Staged Construction Analysis. (It is also possible to apply the equivalent member loads due to stressing a tendon as a load case in a standard static analysis, but we will not cover that here.)

We will create a new construction step at the end of our construction sequence for stressing the tendon.

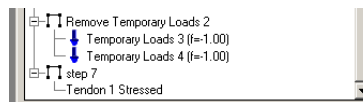
- ◆ Open the Construction Stages Explorer.
- ◆ Click on Stage 1 to activate it, and then click on **Add Step** to add a new step to the end of the stage.
- ◆ Right-click the new step and choose **Tendon Activities**.
- ◆ In the activity spreadsheet, add a new row. Change the Tendon to Tendon 1.



Tendon Stressing Activity

- ◆ Close the activity spreadsheet.

Be sure the activity appears correctly at the end of the Construction Stages Explorer:



Tendon Stressing Activity

Tendon Results

- ◆ Re-run the Staged Construction Analysis. Close the analysis window after it completes successfully.
- ◆ Use deformed model graphical results and step through the construction stages again. Verify that the last step for stressing the tendon exists and that the structure has the expected response.

The “incremental” option in the Graphical Results Options floating window is useful in this case. When this option is on, the results show the changes from the previous step, rather than the cumulative displacements. It can be used with any result type and can be found in the results spreadsheets as well.

Special tools are also available specifically for tendons.

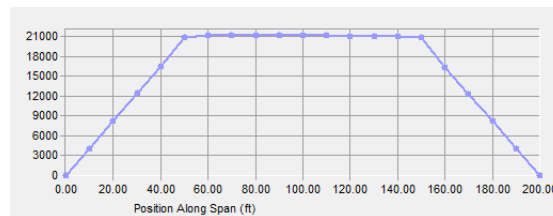
- ◆ Make sure the final tendon stressing activity step is selected in the Analysis Results Explorer.

☛☛ Activate **Results** → **Tendon Results**.

The Tendon Editor will open to the Forces & Losses tab. If this were a Time-Dependent Staged Construction Analysis, the graph would show both short-term as well as long-term losses due to steel relaxation and superimposed loads.

☛☛ Change to the **Primary & Secondary Moments** tab. Then choose Mz.

This graph shows the primary moments in the model along the girder. Secondary moments are also available.



Tendon Primary Moments

☛☛ Close the Tendon Editor.

There are also several tendon-specific results spreadsheets for reporting forces and primary and secondary moments.

☛☛ Go to **Results** → **Spreadsheets** → **Tendon Forces** → **Primary @ Member Ends**.

This spreadsheet reports tendon forces at the start and end of each member along the tendon path, including the tendon tensile force and the primary force in the member local coordinate system directions.

We will add time-dependent material effects in the next section of this manual.

For More Information, please refer to the following documentation.

- For more information on defining tendons, see Tendons in *LARSA 4D Reference Manual*.

Time-Dependent Material Effects

LARSA 4D's Time-Dependent Staged Construction Analysis is a more advanced version of the Staged Construction Analysis that includes time-dependent effects including the effect on concrete elastic modulus, creep, shrinkage, steel relaxation for tendons, and the effects of superimposed loads. The CEB-FIP 78 or CEB-FIP 90 codes are supported, among several others.

The user must choose a code so that the analysis knows which rules to follow, and which input to ask from the user. CEB-FIP 78 is the more flexible code and is based on user-supplied time-property curves for each section, whereas CEB-FIP 90 is equation-based and requires very little input from the user. For this example, we will use CEB-FIP 90.

A time-dependent analysis has the following steps:

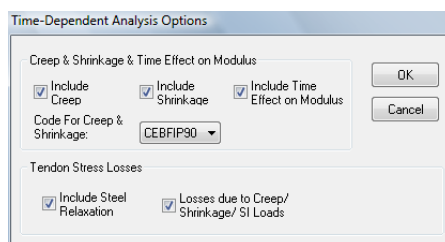
- Choosing the design code.
- Indicate which materials (or sections) are subject to time-dependent material effects and what their time-dependent properties are. No members undergo time-dependent changes unless their material (or section) is explicitly marked as a member having time-dependent effects.
- Setting the casting day on each concrete member that undergoes time-dependent effects.
- Creating the appropriate construction stages at the time points where results are desired.
- Running the analysis.
- Evaluating the cumulative, incremental, and partial results due to time-dependent changes.

Choosing the Design Code

Time-dependent effects must be turned in two ways. First we choose which time-dependent effects to include in the analysis, and according to which code those effects should be computed.

☛ Go to **Analysis** → **Time Dependent Analysis Options**.

☛ Select CEB-FIP 90 as the code, and check all of the options to include all time-dependent material effects in the analysis.



Time-Dependent Analysis Options

Setting Material Properties

We will first indicate which materials in the model are subject to time-dependent behavior. By default, no material is subject to time effects. We will create “Material Time Effect definitions” and assign those to the materials to indicate that they will undergo time effects.

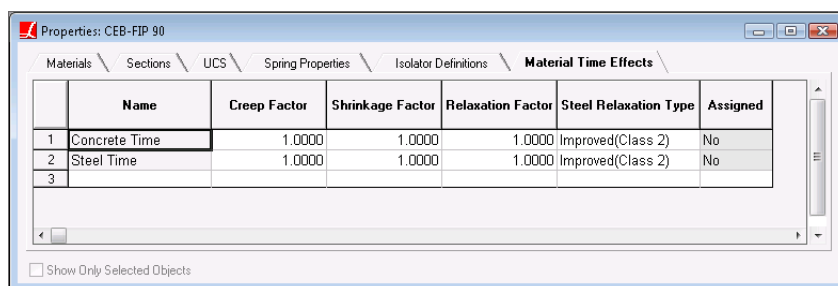
Material Time Effect definitions are used to enter properties such as a creep coefficient and time-versus-elastic-modulus curves. These fields are used for CEB-FIP 78 only, so we will not need to enter them. (Material Time Effect definitions may also be assigned to section definitions in the project if it is more convenient, but a member should not have a Material Time Effect definition on both its material and section.)

- Go to **Input Data** → **Properties** and change to the **Material Time Effects** tab.
- Add one row to the spreadsheet and change the name of the material time effect definition to “Concrete Time”.

The CEB-FIP 90 code provides curves for concrete creep, shrinkage, and the time effect on elastic modulus. It is possible to override the built-in curves through the CEB-FIP 90 menu that shows up when viewing this spreadsheet, but we will not do that here. If CEB-FIP 78 is used or if any of the property overrides are specified, you may need to have separate Material Time Effect definitions for each material in the project. But if the materials share the same time effect settings, you can use the same Material Time Effect definition for multiple materials.

We will need to verify the default curves for stress/GUTS versus relaxation and time versus relaxation for tendons.

- Add a second row to the spreadsheet and change the name to “Steel Time”.



	Name	Creep Factor	Shrinkage Factor	Relaxation Factor	Steel Relaxation Type	Assigned
1	Concrete Time	1.0000	1.0000	1.0000	Improved(Class 2)	No
2	Steel Time	1.0000	1.0000	1.0000	Improved(Class 2)	No
3						

☐ Show Only Selected Objects

Material Time Effect Definition

- Right-click the row (or use the **CEB-FIP 90** menu) and choose **Edit Curve: Stress/GUTS VS Relaxation**. Verify that the default curve is acceptable.
- Close the curve spreadsheet. Right-click the Steel Time row again (or use the **CEB-FIP 90** menu) and choose **Edit Curve: Time VS Relaxation**. Verify that this default curve is acceptable.

The coefficients for the two curves are combined multiplicatively.

- Close the curve spreadsheet window. Go to the materials spreadsheet by clicking the **Materials** tab.
- Change to the **More Properties** subtab.
- For the A992 and Fc_4 materials, change the Material Time Effect column to “Steel Time” and “Concrete Time”, respectively.

Note that the Concrete Cement Hardening Type is set to Normal for this material, versus Not Concrete for the steel material, indicating Fc_4 is concrete and subject to time effects. These options came from LARSA's material database.

- ☛ Close this spreadsheet window.

Casting and Construction Day

Earlier we said that each stage represents a day of construction. For the Time-Dependent Staged Construction Analysis, the day of each stage (which we must set) impacts the material properties of the members and of the tendon. A casting day must also be set for each concrete member undergoing time-dependent effects.

The days entered into LARSA 4D are all relative. We will choose day 10 as the day on which the concrete members are cast (i.e. poured). They will be constructed (i.e. added to the model as load bearing elements) on day 30. Then because we are interested in the time-dependent effects following the final day of construction, we will be interested in the state of the structure on various days up to 5 years in the future.

By default, the casting day of members is day 0. We will modify the casting day on the members spreadsheet.

- ☛ Open the members spreadsheet.
- ☛ Scroll over to the right and change the Casting Day to 10 for all members. It will help to use the spreadsheet edit tool in the toolbar to change all cells at once.

The casting day is ignored for members for which the time-effect on elastic modulus does not apply. In this model, all member elements are concrete and are subject to the time effect on elastic modulus.

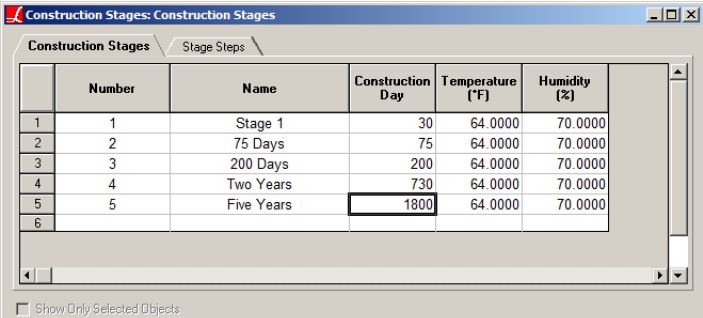
- ☛ Close the members spreadsheet.

The next step is to set the day of construction for the construction stage. The temperature and humidity on this day is also important.

- ☛ Open the Construction Stages spreadsheet from **Input Data** → **Construction Stages**.
- ☛ Change the Construction Day of the stage to day 30.
- ☛ Change the Temperature of the stage to 64 (degrees F) and the Humidity to 70 (percent).

We want to investigate the state of the structure at several time points later on. Each time point to investigate must have a stage, even if the stage has no construction steps or construction activities in it.

- ☛ Add four new stages by adding rows to the spreadsheet.
- ☛ Set their names to 75 Days, 200 Days, Two Years, and Five Years.
- ☛ Set their Construction Days to 75, 200, 730, and 1800.



	Number	Name	Construction Day	Temperature (°F)	Humidity (%)
1	1	Stage 1	30	64.0000	70.0000
2	2	75 Days	75	64.0000	70.0000
3	3	200 Days	200	64.0000	70.0000
4	4	Two Years	730	64.0000	70.0000
5	5	Five Years	1800	64.0000	70.0000
6					

☐ Show Only Selected Objects

Setting Construction Days

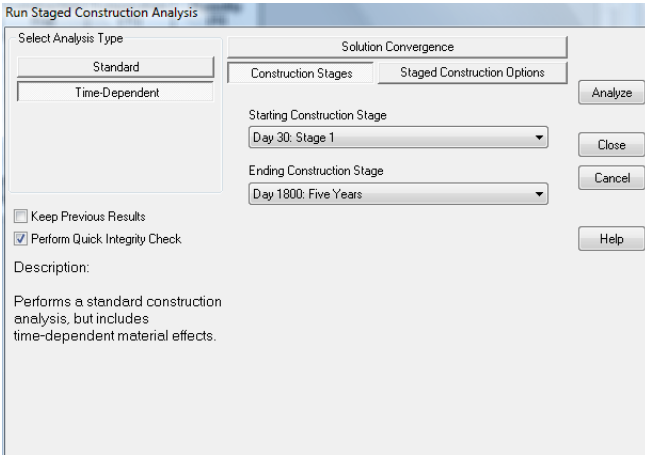
- ☛ Close the spreadsheet.
- ☛ Open the Construction Stages Explorer and note how it has been updated with the new stage data at the bottom.

These stages do not need construction steps because the only activities within these stages are automatically created by the analysis engine.

Running the Analysis

Next we prepare to run the analysis.

- ☛ Go to **Analysis** → **Staged Construction Analysis**.
- ☛ Choose the **Time-Dependent** analysis type.
- ☛ Open the **Construction Stages** options tab, and set Ending Construction Stage to the last stage, Day 1800: Five Years.



Run Staged Construction Analysis

Select Analysis Type

Standard

Time-Dependent

Solution Convergence

Construction Stages Staged Construction Options

Analyze

Close

Cancel

Help

Starting Construction Stage

Day 30: Stage 1

Ending Construction Stage

Day 1800: Five Years

☐ Keep Previous Results

☒ Perform Quick Integrity Check

Description:

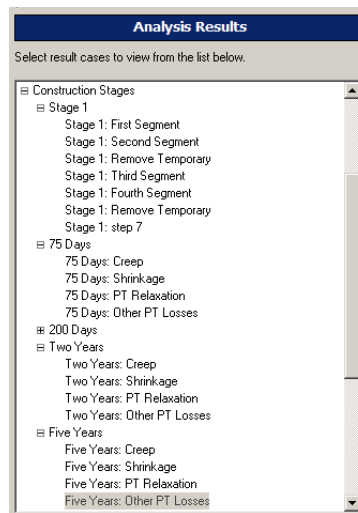
Performs a standard construction analysis, but includes time-dependent material effects.

Staged Construction Analysis Options

- ☛ Click **Analyze**.
- ☛ Close the analysis window when it finishes without errors.

Results

In a Time-Dependent Staged Construction Analysis, new result cases are automatically generated in each stage for the effects of creep, shrinkage, relaxation, and other losses.



Time-Dependent Result Cases

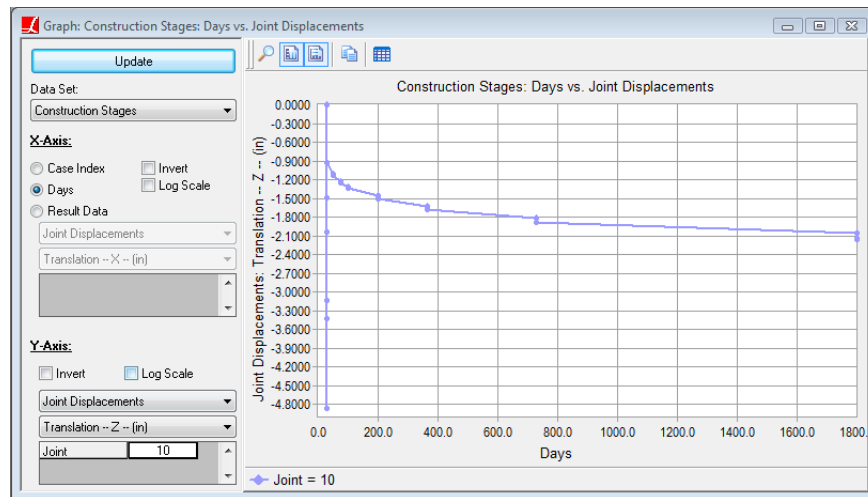
- ☛ Step through deformed model graphical diagrams at a scale factor of 256 from the first result case to the last to see the effects of the time-dependent material properties.
- ☛ If not all of the elements appear deformed, ensure they are selected (use **Select All**) and that the correct stage is selected in the Construction Stages Explorer (use the **View Full Model** button in the explorer if necessary).

The deformations over time are of course also available in the spreadsheets. We can also make a displacement-over-time graph.

- ☛ Go to **Results** → **Graphs**.
- ☛ Change Data Set to “Construction Stages”
- ☛ Change the X-Axis type to Days.
- ☛ Change the Y-Axis result to Joint Displacements and Translation Z and set the Joint number to 10, which is the midpoint of the first span.
- ☛ Click **Update** at the top of the window to draw the graph.

The graph shows that as a result of time, the joint is displacing downward more and more.

The graph shows a stepping function because of the way material time effects are evaluated in each stage. The effects of creep, shrinkage, and other effects are assessed separately within the same day of construction. Each has a small but instantaneous effect, resulting in several steps straight down.



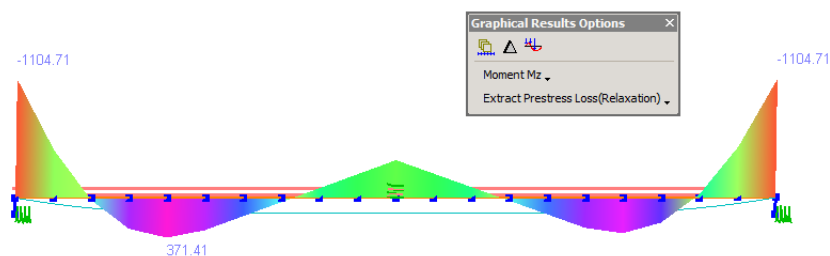
Displacement Graph

- ☛ Close the graph window.
- ☛ Change to moment diagrams and turn on the incremental option to see the effects of time at each stage. Then turn off the incremental option.

We can also display the total effects of static loads, post-tensioning, and each time-dependent effect separately. That is, the cumulative effects of the different types of loading on the structure. LARSA 4D calls this extraction of partial effects by load class.

- ☛ Select the last result case, Five Years: Other PT Losses.
- ☛ In the floating Graphical Results Options floating window, change “View Full Cumulative Results” to “Extract Prestress Loss(Relaxation)”.

This load class is used for long-term post-tension relaxation effects.



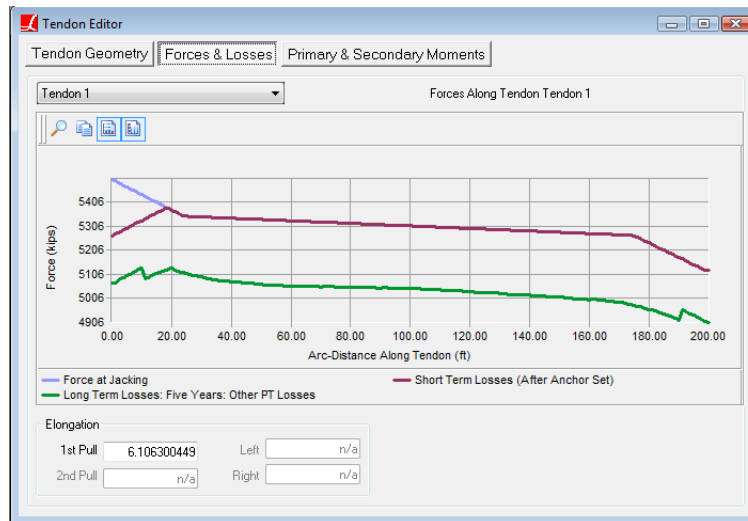
Partial Cumulative Effect for Relaxation

- ☛ Turn off graphical results.

Finally, we can go back to the tendon results to see the changes in the forces within the tendon and primary and secondary moments after long-term losses.

- ☛ Open **Results** → **Tendon Results**.

This graph, which including the short-term losses lines initially, now shows the tendon forces at the selected result case, five years later, in green. You can also use this window to look at primary and secondary moments at this point in time. You must select a result case with post-tensioning (“PT”) activity to get long-term tendon results.



Long-Term Tendon Losses

Haunches

Haunches are added to the model using non-prismatic variation of cross-section geometry and properties. Non-prismatic variation can be added to any parametrically defined cross-section definition, including the custom section shapes and sections defined in LARSA Section Composer.

Non-prismatic variation is created in two steps. First, formulas must be assigned to parameters in the cross-section to determine how the parameters vary over the length of the span. These formulas can define arbitrary curves including linear variation, parabolas, and cubic curves. The second step is to apply the cross-section to the model, possibly using special span definitions so that the variation is applied not just over the length of a single member but across multiple members.

In this example, we will create haunches over the abutments and the pier. The cross-section will decrease in depth from the first abutment over several feet, remain constant, and then shortly before the pier it will begin to increase in depth again. The second span will have the same shape.



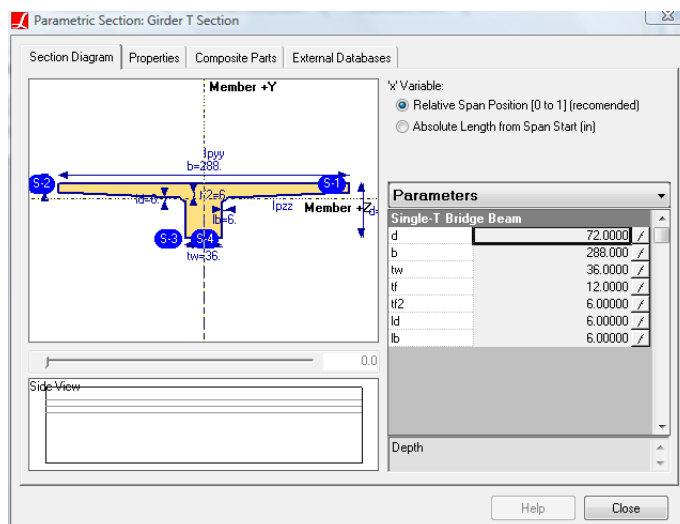
Haunches

Keep in mind when creating non-prismatic variation that member elements are still assumed to have constant properties throughout their length during an analysis. Members must be divided into sufficiently short pieces so that any smoothly curved non-prismatic variation is approximated as close as necessary.

Creating a Non-prismatic Section

Before we create the non-prismatic variation that we want for this model, which involves a complex definition for depth, we will start with a simpler case. Let us have the depth vary linearly from 72 inches at the abutment to 50 inches at the pier. We will modify the existing cross-section definition in the project and add this variation.

- ☛ Open **Input Data** → **Properties** and change to the **Sections** tab.
- ☛ Right-click the T section and choose **Edit Parameters**.



Editing Section Parameters

This command, available for parametrically defined sections only, opens a window which can be used to modify the parameters we initially set for the section.

The T shape section, as with most other custom parametric sections provided with LARSA 4D, is defined with a depth parameter that makes it easy to have the depth of the section vary along the length of a span. When we change the depth parameter, the top surface of the flanges remains in place while the bottom of the web moves up and down — relative to the member reference axes, and thus the rest of the LARSA 4D model.

In this window, we can enter numeric values for parameters, but we can also enter formulas using a special x variable. This variable represents the position on the span. It has the value zero at the start of the span, and depending on how we proceed it can range from 0.0 to 1.0 along the span or it can have actual length values from 0.0 to the length of the span.

To create non-prismatic variation, we enter a formula for the depth parameter d in terms of the x variable.

- ☛ Change the value for the d parameter from “72” to “72 - 22*x”

This is a formula for linear variation. If x varies from 0.0 to 1.0, then the depth of the section will vary from 72 to 50. You can see the preview of the section update at the lower left of this window.

- ☛ Click **Close** to apply the change to the cross-section definition.
- ☛ Go back to the graphics window. Turn on **Graphics** → **Complete Rendering**.

This saw-tooth pattern is clearly not what was desired. The linear variation is repeated within each member! This is because LARSA 4D does not know what the linear variation should range over: a single member, a few members, a whole span, or the entire girder. We must tell LARSA what to do.



The sawtooth pattern is corrected next.

In this case, we will want to segment the girder into the two spans. In the final setup for the haunches, we will have the non-prismatic variation apply span-by-span, that is, repeating after the pier.

Spans

To tell LARSA 4D what members the variation should range over, we create what are called “spans”. These spans in LARSA 4D do not necessarily need to correspond to a span in the bridge, e.g. from pier to pier — they are just a tool to identify a range of contiguous member elements. In this case, though, we do want the spans to match the two spans in the model.

The easiest way to create spans is to select the members that form a single span and then use a tool to create a span from the selected members.

- ☛ Turn **Unselect All**, and then select just the members in the first span. It may help to turn **Complete Rendering** on and off in this process.

If you turn **Complete Rendering** off, you’ll see that the members are drawn in the wireframe diagram also with something going on. When **Complete Rendering** off is off, the members are drawn at their centroids. Because of the non-prismatic variation, the member lines no longer touch the joints perfectly as the location of the centroid varies along the member.

- ☛ Go to **Draw** → **From Selected Geometry** → **Span**.
- ☛ Have spans be included in the graphical display of the structure by opening **Graphics** → **Show** and making sure **Spans** is checked.

The span will be depicted by a bolder green line through the members in the span.

- ☛ Turn **Complete Rendering** back on.

The linear variation has been reworked to range across the entire span, rather than repeating member-by-member.



The linear variation now correctly runs the length of the first span

- ☛ Make a span for the second span: Unselect All, then use the **Draw** menu tool again.

In the second span the linear variation is probably going in the wrong way for what would make the most sense. To reverse the direction of the variation on just the second span, a second cross-section definition would need to be created with a different formula. (The direction of the member elements could also be reversed in the second span, but we recommend keeping all member local axes pointed in the same direction so that the sign convention for results is consistent throughout the model.) However, in the design we actually want the variation will be symmetric within each span, so we will not need to create a second cross-section.

In the next step we will revise the variation so that we get haunches at the abutments and pier.

The next time you save the project, you will be asked to recompute the torsion constant J . This constant needs to be recomputed now for each member because the cross-section properties now differ in each member.

Simple Parabolic Variation of Depth

To create the haunch described at the start of this section, consider the variation that we want within a span. At first there will be parabolic variation in depth from 72 inches to 50 inches over 20 feet. Over the next 60 feet the depth will remain constant at 50 inches. Then from 80 feet to 100 feet the parabolic variation will repeat in reverse, from 50 to 72 inches. This requires us to define a piecewise variation function.

Before we create a piecewise definition, we will apply a simple parabolic function that has the depth vary from 72 inches at the sides to 50 inches mid-span.

- ◆ Return to the **Edit Parameters** window by going to the sections spreadsheet and then right-clicking the section row.

(You may notice that in the spreadsheet the section no longer displays its properties. This is because the properties vary depending on position.)

We could revise the formula here and enter a second-order polynomial ourselves. However, LARSA 4D has a formula helper tool that makes it easier to construct complex formulas for non-prismatic variation.

- ◆ Click the f (for “formula”) next to the d parameter field.

The formula helper opens with a window to change the linear variation. Rather than typing in a formula from scratch, the helper has fields for the start (72) and end values (50). It will create the appropriate formula.

- ◆ Change to the **Parabolic** tab. Enter a start value of 72, an initial slope of -88, and a final slope of 88.
- ◆ Click **OK**.

The side-view preview diagram will update to show the curve. The parameters spreadsheet will also show the actual equation in x used. You can substitute 0.5 for x to verify that the depth at mid-span is 50 inches. Note that the initial slope of -88 refers to the fact that depth is *decreasing* at the start. The slope is in depth units (inches) per units of the x variable, which ranges from 0.0 at the start to 1.0 at the end of the span.

- ◆ Click **Close**.

Verify that the complete rendering diagram in the graphics window shows the haunches properly.

Piecewise Variation

Now we will create the piecewise variation function.

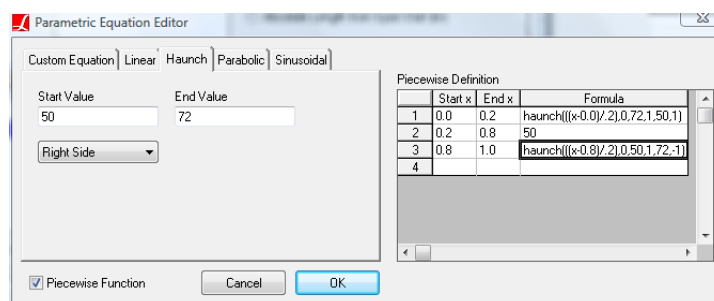
- ◆ Return to the **Edit Parameters** window by going to the sections spreadsheet and then right-clicking the section row.
- ◆ Click the f (for “formula”) next to the d parameter field.
- ◆ Click the **Piecewise Function** checkbox at the bottom of the window.

A spreadsheet opens to the right where we can configure a complex piecewise function.

- ☛ In the first row in the piecewise definition spreadsheet, enter 0.0 for start x and 0.2 as end x.

We are treating the x variable as ranging from 0.0 at the start of the span to 1.0 at the end. This condition refers to the first 20% of the span, or 20 feet.

- ☛ Change to the **Haunch** tab. Enter a start value of 72 and an end value of 50.
- ☛ In the last, blank row in the piecewise definition spreadsheet, enter 0.2 for start x and 0.8 as end x.
- ☛ In the Custom Equation field, just enter “50”.
- ☛ In the last, blank row in the piecewise definition spreadsheet, enter 0.8 for start x and 1.0 as end x.
- ☛ Change to the **Haunch** tab. Enter a start value of 50 and an end value of 72. Change the “Left Side” option to “Right Side”.



Creating piecewise variation in depth

- ☛ Click **OK**.

The side-view preview diagram will update to show the haunches.

- ☛ Click **Close**.

Verify that the complete rendering diagram in the graphics window shows the haunches properly.

- ☛ Re-run the analysis with the new cross-section properties and inspect the results.

For More Information, please refer to the following documentation.

- For more information on creating non-prismatic variation, see Nonprismatic Variation in *LARSA Section Composer Manual*.
- For more information on the LARSA formulas language, see Parameters and Formulas in *LARSA Section Composer Manual*.

Index

AASHTO LRFD, 23
bearings, 11
break members, 11
bridge example,
 casting day, 47
CEB-FIP 78, 90, 47
combinations, 23
construction loads, 33
dead load, 17
deformed model, 19
drawing members, 11
example,
 grounded springs, 11
groups, 33
GUTS, 47
haunches, 55
influence line example, 23
jacking a tendon, 41
joints spreadsheet, 11
linear static analysis, 19
live load example, 23
non-prismatic variation, 55
post-tensioning example, 41
refining a model, 11
results, 19
scale factor, 19
self weight, 17
staged construction analysis example, 33
stress versus GUTS, 47
stressing a tendon, 41
structure groups, 33
temporary loads, 33
tendon editor, 41
tendons, 41
time-dependent material effects, 47
training manual,